Instruction Report GL-97-1
March 1997

US Army Corps of Engineers
Waterways Experiment Station

User's Manual for STUBBS: A Finite Element Program for Geotechnical Analysis

by John F. Peters, Ronald E. Wahl, Ronald B. Meade

19970416 010

Prepared for U.S. Army Corps of Engineers

PRINTED ON RECYCLED PAPER

Instruction Report GL-97-1
March 1997

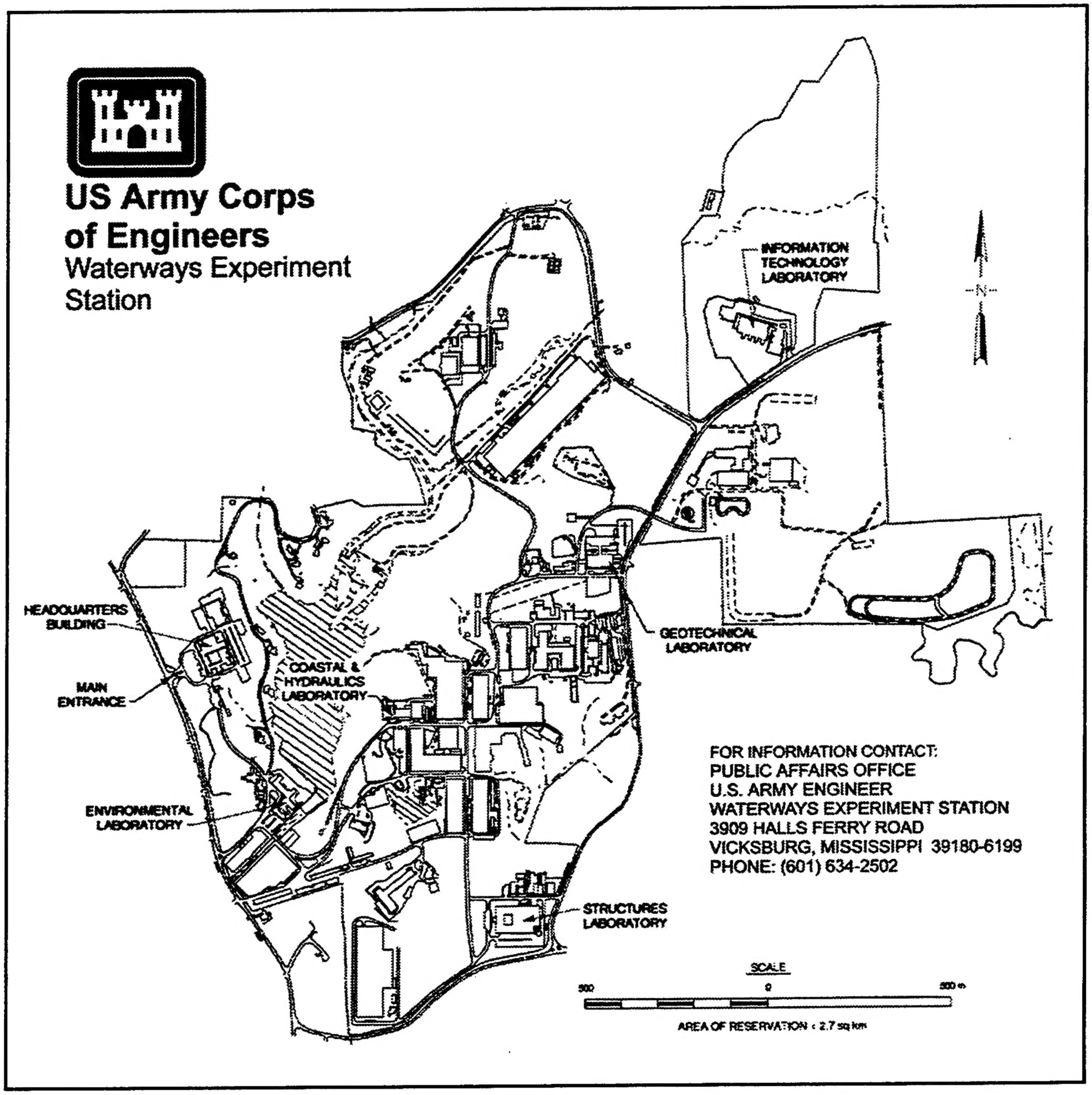
User's Manual for STUBBS: A Finite Element Program for Geotechnical Analysis

by John F. Peters, Ronald E. Wahl, Ronald B. Meade

U.S. Army Corps of Engineers
Waterways Experiment Station
3909 Halls Ferry Road
Vicksburg, MS 39180-6199

Final report

Prepared for U.S. Army Corps of Engineers
Washington, DC 20314-1000



Waterways Experiment Station Cataloging-in-Publication Data

Peters, John F.
User's manual for STUBBS : a finite element program for geotechnical analysis / by John F. Peters, Ronald E. Wahl, Ronald B. Meade ; prepared for U.S. Army Corps of Engineers.
113 p. : ill. ; 28 cm. — (Instruction report ; GL-97-1)
Includes bibliographic references.
1. STUBBS (Computer program) 2. Finite element method — Computer programs. 3. Soil mechanics — Computer programs. I. Wahl, Ronald E. II. Meade, R. (Ronald B.) III. United States. Army. Corps of Engineers. IV. U.S. Army Engineer Waterways Experiment Station. V. Geotechnical Laboratory (U.S. Army Engineer Waterways Experiment Station) VI. Title. VII. Title: A finite element program for geotechnical analysis. VIII. Series: Instruction report (U.S. Army Engineer Waterways Experiment Station) ; GL-97-1.
TA7 W34i no.GL-97-1

Contents

List of Figures

List of Tables

Preface

The U.S. Army Engineer Waterways Experiment Station (WES), under the sponsorship of the Civil Works and Repair, Evaluation, Maintenance, and Rehabilitation Research Programs, developed a finite element program for geotechnical analysis.

The creation of the computer program was primarily the concept of Dr. John F. Peters, Geotechnical Laboratory (GL). Mr. Ronald E. Wahl, Soil and Rock Mechanics Division, GL, was also a principal in the development of the program, and Dr. Ronald Meade was responsible for the preparation of this users manual.

The project was under the direct supervision of Mr. Robert D. Bennett, Chief, Soil Research Facility, and under the general supervision of Dr. Don C. Banks, Chief, Soil and Rock Mechanics Division, and Dr. William F. Marcuson III, Director, GL.

At the time of publication of this report, the Director of WES was Dr. Robert W. Whalin. The Commander was COL Bruce K. Howard, EN.

1 Description of STUBBS

Introduction

STUBBS is a two-dimensional (2-D) finite element code developed for the purpose of solving problems in geotechnical engineering. The code is capable of performing a fully coupled Biot analysis (Biot 1941) in which the partial differential equations for static equilibrium and flow continuity are solved simultaneously. The code can also handle either pure deformation or pure flow problems. The code accounts for nonlinear stress-strain behavior of soils and groundwater flow through both saturated and partially saturated soils. The program is able to account for soil-structure interaction effects using a library of specialized elements. Thus, STUBBS has the potential to deal with soil-structure interaction, consolidation, and transient seepage problems. The program can perform analysis of problems in either plane strain or axisymmetric geometries.

A particular strength of the program is the ability to simulate construction processes. As such, the program can predict movements, displacements, and porewater pressures which might exist at different times during construction. Activities which can be simulated by STUBBS include the excavation and placement of fill, the addition/removal of structural elements such as walls, anchors or bracing, and the placement and removal of distributed and concentrated loads. Analysis of the construction process enables the design engineer to evaluate the effects of construction rates and the sequence of the construction activities. Typical applications of this feature include the construction of embankment dams, lock walls, reinforced earth projects, and sheet pile walls. The simulation of construction activities is controlled by the use of a construction script. A useful reference for the application of the finite element method in geotechnical engineering is Technical Letter No. 1110-2-544 entitled "Geotechnical Analysis by the Finite Element Method."

The STUBBS code uses small strain theory. The limitations on accuracy posed by small strain theory are most important in problems involving time dependent loadings on highly compressible materials. Another version of the code based on large strain theory is currently being developed at WES.

STUBBS was developed at the Waterways Experiment Station (WES) by Dr. J. F. Peters and his colleagues in the Soil and Rock Mechanics Division

of the Geotechnical Laboratory under the Civil Works Research and Development Program and the REMR Program. The computer program was written in FORTRAN77 and runs successfully on several platforms which include personal computers, engineering work stations, and mainframe computers.

Organization of Report

This user's manual is intended to provide the basic information to enable a new user to prepare the files necessary to execute the finite element program STUBBS. A pre- and postprocessor (XMESH) has been developed for STUBBS. The problems presented in this manual do **not** require the use of XMESH. All of the example problems were prepared on a PC using a hand-drawn mesh. A manual for XMESH is in preparation.

This manual can be used in a variety of ways. For the new user, it is recommended that the body of the text be read before attempting to use the program. Then, Appendix B should be read to gain an understanding of the required input file structure. Finally, the user should examine the examples presented in Appendix C. The first example contains the most detail on the data file and shows a complete listing of the output file.

The Keyword Dictionary (Appendix A) and the example problems (Appendix C) provide details to enable the user to prepare files to solve a wide variety of geotechnical problems. A cross reference of commonly used keywords with examples is provided in Table C3 of Appendix C. The cross reference list should enable the user to find examples of keywords contained within data files. All of the examples data files have been provided on disk. A sample FEA_FILE (see Appendix B) is also included.

The Keyword Dictionary provides a complete listing of all Keywords with examples of proper syntax. The dictionary is intended to be the comprehensive reference on STUBBS Keywords.

Program Overview

The STUBBS program consists of a group of subroutine modules. Some modules prepare the user's input for calculations, other modules solve equations of equilibrium, or produce plot files and output files. The aim of the program is to free the user from concern about the detail of the FEM method, and instead, allows the user to focus on the analysis.

The user must provide a data file. The data file is composed of information grouped using keywords. The keywords allow the information to be assembled in a fashion that is relatively insensitive to the order in which the information is entered. The data file is an ASCII file that can be prepared using any text editor. The information is grouped into problem geometry,

material properties, and loading information. An optional fourth category is initialization data, useful in problems involving fluid flow and nonlinear material response. A brief discussion of the data file structure is presented in the following subsections.

Problem preliminaries

The data file starts with a line giving the problem title. Preliminary data include the type of problem (pure deformation, pure flow, or coupled flow) and the system of units.

The finite element mesh

The problem geometry is structured as a mesh composed of nodes that are connected to elements. Several type of elements can be used in STUBBS. Solid elements can simulate soil, rock, concrete, etc. Beam elements can simulate steel, plastic, wood, or any material whose flexural properties control deformation and displacements. Bar elements can simulate steel, concrete, wood, geotextiles, or any material governed by its uni-axial properties. Slip elements simulate joints, cracks, or discontinuities.

Mesh preparation is best done using a preprocessor, although a simple mesh can be hand-generated. A preprocessor called XMESH is available from WES for use with STUBBS. In addition to the node location, each node has three degrees of freedom (X direction, Y direction, and pore water pressure). Each element is composed of a group of nodes and the element is assigned a material description.

Material properties

Each material has a group of mechanical properties. The mechanical properties for solid elements include linear elastic or elastic-plastic properties. Permeability is also a required input parameter for solid elements. Bar elements and beam elements have linear elastic properties. Slip elements have strength and compressibility properties.

Boundary conditions

Point loads, distributed loads, displacements, and pore water pressures can be specified for nodes. Also, material may be added as fill or excavated at any time in the problem. Material properties can be changed at any time. The acceleration of gravity can be altered to simulate the process of centrifuge testing.

Initial conditions

Initial values of stress, fluid pressure, and displacement can be specified. In problems involving flow, the ability to specify initial fluid pressures allows the user to start the problem with conditions that correspond to field measured values.

A summary of the user-supplied data is shown in Table 1. The sections of a typical data file for a pure deformation problem are shown in Table 2. Comment lines may be included in the data file. Comment lines are optional and do not effect the solution. They are particularly useful in identifying specific types of information in the file. Each of the data files shown in this report and provided on the disk accompanying this report use comment lines to identify data types and provide additional information regarding the problem.

Table 1
User-Supplied Information

Information Type	Source	Typical Key Words
Preliminaries	User	SPEC UNIT
Geometry (mesh)	Pre-processor or hand generated	NODE ELEM
Material properties	Lab tests or handbook values	SMPR
Loadings	Design loadings, construction loadings	CSBL CSPL
Initial conditions	Field measured values	IDPP

The information in the data file may be presented in any order so long as the first line in the file is the title and the construction script is begins with by the KW LFTC and the last line of the script is KW DONE. For example, the order of the node data (KW NODE) and the element data (KW ELEM) could have been reversed from that shown in Table 2. For example, the material properties data (KW SMPR) could have been placed between the element and node data.

Element Types

Four types of elements are available to the user to model the behavior of the various types of system components. These are (a) solid elements, (b) bending (beam) elements, (c) bar elements, and (d) slip-separation elements. Typical applications for each of these elements are listed in Table 3.

Table 2
Sample Data File Entries

Information type	Data File Entry
Problem title and comments	STUBBS Hand Mesh # 2 4-node elements C July 25, 1995 C filename = FOURS1.DAT C example using zero unit weight and 10 psi boundary load along top C of specimen (plain strain)
Units and problem type	KW UNIT C ATMOSPHERIC PRESSURE, GAMMA H2O, VALUE OF 1-g (3 required parameters) C psi, pci, units of g 14.7 0.0361 1.0 C KW SPEC C FLOW, DFORM, DRAIN, CHECK_MESH, CHECK_SCRIPT, AXIS (6) F T T F F F
Geometry	KW NODES C Scaling parameters X-factor, Y-factor 1.0 1.0 C Node X-coord Y-coord X Y PWP dof's 1 0.00 0.00 1 1 0 2 1.0 0.000 0 1 0 . 34 3.000 12.000 0 0 0 35 4.000 12.000 0 0 0 C KW ELEM C 4-node connectivity table C Ele# Mat # 4 corner nodes mid-side nodes 1 1 1 2 7 6 0 0 0 0 2 1 2 3 8 7 0 0 0 0 3 1 3 4 9 8 0 0 0 0 . 23 1 28 29 34 33 0 0 0 0 24 1 29 30 35 34 0 0 0 0
Material properties	KW SMPR C linear elastic material properties C MAT#, GAMMAT, RVOID, GS, RSAT (5) 1 0.00 0.55 2.7 .80 C MODULUS, POISX, POISY, XN, XM, KO (6) 1000 .3333 .3333 1.0 0.4 0.5 C PERMX, PERMY, B-Value (3) .001 .001 .999 C van Genuchten parameters (2) 0.006 2.3 C End of material properties
Loadings	KW LFTC C Start of construction script KW TIME C # of substeps, duration 1 1. KW CSBL C Distributed load C Nodes Pressure 31 0 32 -10 -10 -10 32 0 33 -10 -10 -10 KW DONE KW DONE

Table 3
Typical Applications of the Different Element Types Used In STUBBS

Element Type	Applications Element is Used to Represent
Solid	1. Soil including fills and foundation 2. Mass concrete structure 3. Rock
Bar	1. Reinforcement in reinforced earth structures 2. Tie backs for anchor walls 3. Springs 4. Structural Braces/Struts
Beam	1. Flexural members in a building frame 2. Columns in a building frame 3. Sheet pile walls
Slip-separation	1. Interface between soil and rock 2. Interface between fill and concrete retaining wall 3. Interface between soil and reinforcement in reinforced earth structures 4. Rock joints/fractures

Solid elements are used to represent mass materials such as soil, rock, and massive structures. Bar elements are used in situations where a member which can take either tensile or compressive loads is required. The bending elements in STUBBS can carry both axial and transverse loads and are used in situations requiring beam-column type behavior. The slip-separation (interface) elements can be used to simulate the relative movement which can occur at the junction of different materials or where a discontinuity exists in a solid material. The solid element is the only type of element that includes both pore pressures and nonlinear behavior. The mechanics of solid elements are discussed in detail in Chapter 2. Slip-separation elements are nonlinear as are tensionally or compressionally only bars. Further details on the solid, bar, beam and slip-separation elements are discussed in Chapter 3 of this report.

Keywords

The program input is structured using keywords (KW). Keywords are used as flags that are used to direct data into the appropriate arrays and call subroutines. A complete list of keywords is provided in APPENDIX A. The preparation of an input file is discussed in detail in APPENDIX B. A detailed description of the KW SPEC is provided in the following paragraphs. A discussion of the other KW's are provided in the Appendices.

As mentioned previously, the code can handle either (a) coupled deformation/flow, (b) pure flow, or (c) pure deformation problems. The selection of these conditions is controlled by the state of LOGICAL VARIABLES which are set in the input file using the keyword SPEC. Six logical variables are set by the user in the input data. These logical variables are

used to set flags that control the manner in which subroutines are called. The variables are set to either true "T" or false "F" on the data line following the KW SPEC line.

KW SPEC initializes the logical variables which describe the type of problem to be solved. These logical variables pertain to flow, deformation (both coupled and uncoupled), drainage conditions, and whether the problem is plane strain or axisymmetric. Also, KW SPEC contains information which specifies whether the mesh and construction script are to be checked.

The six logical variable states are placed on one line separated by spaces. The variable states (T or F) must appear in the following order:

(one line only) FLOW, DFORM, DRAIN, CHECK_MESH, CHECK_SCRIPT, AXIS

If FLOW is T, then flow (continuity) equations are included in the finite element solution. If FLOW is F, then the flow equations are excluded from the finite element solution. All flow problems are time dependent (transient), however, the steady state solution can be obtained by maintaining constant boundary conditions for a sufficiently long period of time. The solution of time dependent problems requires the use of a time step to be specified in the input to the flow problem. The KW TIME in the construction script specifies the length of the time increment during the current time step and also the number of substeps that are used within a time increment. The KW 'TIME' is required when modeling all time dependent processes such as consolidation, transient flow, temperature change, and creep with STUBBS. A discussion of time dependent problems is presented in Chapter 4.

If DFORM is T, then deformations are to be included in the finite element solution. If DFORM is F, then the calculation of displacements are excluded from the finite element solution.

If both FLOW and DFORM are T, the program solves the equation system for coupled flow. If only one of the two variables, FLOW and DFORM, is set T, then the problem is solved as either pure flow (FLOW =T, DFORM =F) or pure deformation (FLOW= F, DFORM =T). Clearly, both of the variables should not be set to false.

If DRAIN is T, then the drained conditions are to be modeled by STUBBS. If DRAIN is F, then the fluid bulk modulus is added to the stiffness of the solids.

CHECK_MESH and CHECK_SCRIPT are variables whose state indicates if the data input file will be checked for errors before expending time or computer resources on a solution. If either of these variables is set to T, the program input data will be checked and the results of the check reported in the error file. No solution is attempted regardless of the outcome of the check.

If CHECK_MESH is set to T, then the input data are checked for its completeness and STUBBS does not provide the finite element solution. The output file contains a report of the input from the data file. The report will not include the mention of construction sequences regardless of the state (T or F) of CHECK_SCRIPT.

If CHECK_MESH is set to F and CHECK_SCRIPT is set to T, the output file report includes a description of the construction sequence. If both CHECK_MESH and CHECK_SCRIPT are set to F, the program will attempt to produce a solution after the data file is read.

The logical variable AXIS permits the stiffness matrix calculations to model axisymetric elements when set to T. When the variable is F, the elements have a plain strain configuration.

Simulation of Construction

The program is designed to permit elements to be added, removed or altered to simulate construction processes such as filling, excavating, and reinforcing or modifying soil. The construction activities are controlled by keywords that form a construction script. All construction script keywords begin with the letter C. For example, the KW CSFP indicates that a list of solid elements are to be added during the current time step. The KW CSEX indicates that a list of solid elements are to be excavated during the current time step. The KW LFTC is required for all problems. The KW LFTC marks the beginning of construction sequences and data. Details of simulating the construction process by means of a script are presented in Chapter 4.

Programming Conventions

Sign conventions

The sign convention for direction of point loads and displacements is that the upper right quadrant on a Cartesian plane is positive. The sign convention used for applying distributed loads using KW CSBL depends on the order in which the node numbers are entered. A complete discussion of the CSBL sign convention is presented in Appendix C. The Cartesian plane is labeled in terms of X and Y coordinates. The Y-direction is vertical, the X-direction is horizontal, and the Z-direction is normal to the X-Y plane. Again, to the right is positive for X and up is positive for Y. When solving axisymmetric problems, the Cartesian plane is the R-Y plane. That is, the X-axis is the radial or R-axis. No negative values of R are permitted. The R=0 line is the axis of symmetry.

The sign conventions influence both direction of deformation and the direction of loadings. For example, in a plane strain problem, a downward

acting foundation loading acts in the Y-direction and downward displacement has a negative sign.

The sign convention for normal stresses is that compression is reported as positive and tension as negative. Shear stress is reported as positive if the couple on the X-face tend to rotate the element in a counter-clockwise direction.

Units

The program can use any consistent system of units. Length and force units are implied by the values of atmospheric pressure and unit weight of water specified using KW UNIT. The units of time must be consistent with the choice of permeability units in KW SMPR. Also, the parameters used within the program were selected for ease of computation and vary somewhat from the parameters used in conventional geotechnical engineering. For example, the compression index is not an input parameter in coupled flow (consolidation) problems. A partial list of parameters used in linear elastic problems is shown in Table 4. Those parameters and associated KW's that must be provided in every problem where solid elements are used are shown in **bold**. Permeability parameters must be provided in every problem using solid elements, although the parameters are used in the solution for flow and coupled flow-deformation problems only.

The fundamental dimensions used in STUBBS are force (F), length (L), and time (T). Typical parameters and the associated KW's are shown in Table 4.

Table 4
Typical Parameters and Associated KW's

Parameter (Dimensions and Usage)	Associated KW's
F - concentrated loads	CSPL
L - **coordinates of nodes**, displacements	**NODE**, CSBC
L^2 - area of beam elements	BEND
L^4 - moment of inertia for beam elements	BEND
T - time step duration	**TIME**
F/L^2 - atmospheric pressure, van Genuchten parameter α (see Chapter 2), **elastic modulus**, distributed loads, initial stresses, pore water pressures	**SMPR**, CSBL, ISRE, CSBC
L/T - permeabilities	**SMPR**
F/L^3 - unit weight	**SMPR**

File name conventions

The program STUBBS uses seven files during the course of execution. One of the seven files called FEA_FILE holds the names of the other six files. The program creates a mesh file, an error log file, a plot file, an output file, and a file that contains bar and beam element data. Details regarding these files and instructions for preparing the input and FEA_FILE are provided in Appendix B.

2 Material Behavior of Solid Elements

Solid elements are typically used to represent soil and rock. The behavior of solid elements in STUBBS is generally based on the constitutive relationships of the geological materials. The properties for the behavior of solid elements are divided into two distinct categories. Those controlling deformation and those controlling flow. The properties used during an analysis are controlled by logical variables which define whether or not the problem is coupled deformation/flow, pure flow, or pure deformation. The following sections describe the behavior of the solid elements for both deformation and flow.

Deformation Properties

Deformation behavior can be divided into two separate categories: linear elastic and elastic-plastic (nonlinear). Every element in the mesh is assigned to a material type. A particular type of material may have linear elastic and/or nonlinear material properties. Thus, in a given mesh, it is possible for some elements to be linear elastic and other elements to be nonlinear. Deformation properties come into play only when the logical variable for deformations is set to true.

Linear elastic materials

The constitutive law for linear elastic materials is Hooke's law. The material parameters required by STUBBS for linear elastic solid elements include Young's Modulus, Poisson's Ratio, the at-rest earth pressure coefficient, and saturated permeabilities along with two parameters describing the soil suction permeability relationships for partially saturated soils. The two soil suction parameters are called van Genuchten parameters and are discussed in detail in a following subsection. In addition, anisotropy may be considered by providing different flow and compressibility parameters for horizontal and vertical directions.

Nonlinear materials

The program has the capability to model nonlinear plastic behavior using an endochronic model developed by Valanis and Peters (1991) and adapted for use in STUBBS by Peters (e.g. Peters and Valanis 1993). The nonlinear material properties are input using the keyword ENDO.

Flow Properties

Flow properties become a factor in the problem if the logical variable for flow is switched on. Flow through porous media falls into two regimes: (a) fully saturated, and (b) partially saturated. Additionally, STUBBS can model the flow of groundwater through soils having anisotropic permeabilities.

The flow of water through porous media obeys Darcy's law, thus in STUBBS, the soil permeabilities must be provided as input. Since anisotropy is considered, these permeabilities are those in the x and y directions, k_x and k_y. If the analysis proceeds in the pure flow mode, then it is assumed that the ground water flow occurs though a rigid soil skeleton and no attempt is made to account for the soil's potential to shrink or swell. On the other hand, when deformations are considered, the governing differential equation accounts for shrink/swell potential of the soil in the coupled mode as computed changes in pore pressures result in corresponding changes of effective stress, which in turn result in volume changes in the soil.

The governing differential equation for ground water flow in STUBBS is *Equation (1)* listed below:

$$k_x \frac{\partial^2 h}{\partial x^2} + k_y \frac{\partial^2 h}{\partial y^2} = c \frac{\partial h}{\partial t} \tag{1}$$

where

k_x = horizontal permeability

k_y = vertical permeability

C = storage coefficient

h = total head

In *Equation (1)*, the storage coefficient, can be expressed as:

$$c = \frac{\delta\theta}{\delta h} \tag{2}$$

where

θ = volumetric water content

θ = nS

S = Degree of Saturation

n = Porosity

Note that in steady-state flow problems, the storage coefficient on the right-hand side of *Equation (1)* is zero and becomes Laplace's equation. In partially saturated flow, C is proportional to the reciprocal of the slope of the suction-saturation curve.

Since non-steady state flow problems are time dependent, during each time step, the permeabilities, k_x and k_y, are determined based on the current degree of saturation of that element. If the element has a saturation of 100 percent, the element is assigned values of k_x and k_y corresponding to the saturated permeabilities, $(k_x)_{sat}$ and $(k_y)_{sat}$. The saturated permeabilities are material parameters which are input directly to the program.

If the element is partially saturated, its saturation is determined through the use of a suction-saturation relationship for that material. A soil-suction curve for a partially saturated soil is shown in Figure 1.

In STUBBS, this relationship is defined using a modified version of the van Genuchten formula (van Genuchten 1980) shown as *Equation (3)*. The van Genuchten parameters are not fundamental parameters. Instead, they are parameters used to fit a curve which matches experimental data.

$$\overline{S} = (1 + |\alpha h|^{\beta})^{-m} \tag{3}$$

where

h = suction or negative pore water pressure in psf

β = van Genuchten parameter

α = van Genuchten parameter in units of 1/h, i.e., sq ft/lb

and

$$m = (1 - \frac{1}{\beta})$$

The modification to the van Genuchten formula is in the suction term, h. In the expression provided by van Genuchten (1980), h was in terms of head and had dimensions of length, typically cm. In the modified formula the suction is in terms of pressure, i.e., kN/m^2. The product term, αh, is dimensionless so α must have dimensions of 1/h. Also, van Genuchten used volumetric water content on the left-hand side of the expression instead of the relative saturation as shown in *Equation (3)*. The replacement of volumetric water content by relative saturation does not alter the form of the equation.

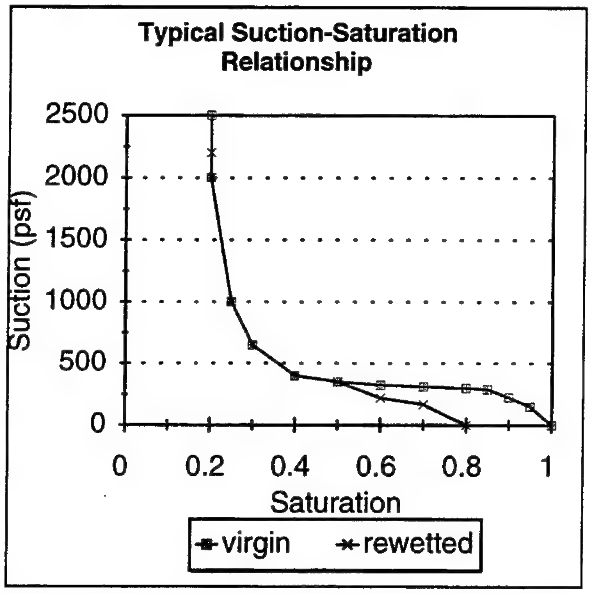


Figure 1. Typical soil suction relationship

In Equation (3), S, the relative saturation, is defined by *Equation (4)* below:

$$\bar{S} = \frac{(S - S_r)}{(S_S - S_r)} \tag{4}$$

where

S = current value of degree of saturation

S_r = residual saturation

S_s = degree of saturation if soil is fully saturated

S_s = 1 (typically)

Values of α, β, and S_r are required input parameters for each solid element material in the finite element mesh. These parameters are necessary to define the suction-saturation relationship using the van Genuchten model. Typical values of these parameters are listed in Table 5. Suction-saturation curves derived from the properties in Table 5 are presented in Figure 2.

Table 5
van Genuchten Parameters for Typical Soils

Soil type	α (units, 1/h) (1/psf)	β	S_r
Sand	0.0083	6.5	0.17
Silt	0.0022	5.0	0.20
Clay	0.0035	2.0	0.84

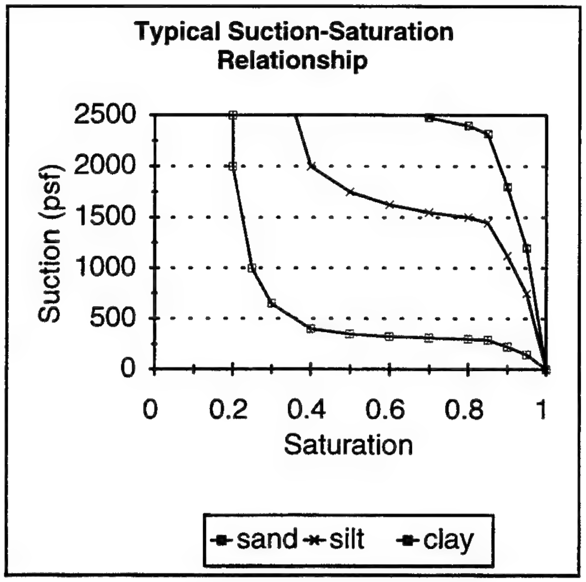


Figure 2. Soil suction for various soils

The van Genuchten relationship given in Equation (3) represents a drainage suction-saturation curve derived from tests performed on samples that are initially fully saturated. In these types of tests, the water content is decreased by applying a suction to the sample. The suction is increased until the residual saturation level is reached. At this level of saturation an increase in suction produces no decrease in water content. Data show that if the sample is "rewetted" the data points do not remain on the drainage curve. In this version of STUBBS these hysteretic effects are not considered and all points were assumed to lie on a unique drainage curve. The hysteretic effects are shown in Figure 1.

Permeabilities are determined using the relationship between relative saturation and the partially saturated permeability, k, listed in *Equation (5)*.

$$k = k_{sat} \times \overline{S}^3 \qquad (5)$$

The permeability of partially soils is a nonlinear material property since the relationship between suction (negative porewater pressure) and saturation is nonlinear. In *Equation (3)*, the unknowns are total heads, however, in the solution algorithm for STUBBS, because the gravity (elevation) head is known, it appears as a "force" and STUBBS actually solves for the pore pressures (pressure head). The porewater pressures at each nodal point are based on the current saturation levels and permeability for each element in the embankment-foundation system. The nodal porewater pressures are interpolated to the Gauss integration points of each element. The elemental porewater pressures are used to determine whether or not the element is

saturated (based on whether the value is positive or negative). If the pore pressure is positive, the saturation is 100 percent and the saturated permeability is assigned to that element for the next time-step. If the pore pressure is negative (suction), the soil is considered partially saturated and the saturation level is determined from *Equation (4)* and the permeability is determined from *Equation (5)*. At present STUBBS does not perform interactions for partially saturated problems. Rather, properties for a time-step correspond to the values assigned at the end of the previous time-step. It follows that small time steps should be used when there is significant movement of the phreatic surface.

3 Finite Element Library

As mentioned in the introduction of Chapter 1, four basic types of elements are employed within STUBBS. These include: (a) solid, (b) bending, (c) bar, and (d) slip-separation elements. All elements are 2-D and the solid elements can be used in either plane-strain or axisymmetric coordinate systems. The following paragraphs give brief descriptions of the essential features of each type of element.

Solid Element

The basic solid element employed by STUBBS is presented in Figure 3. This element is an isoparametric quadrilateral which uses four point Gaussian integration in the computation of the element stiffness matrix. Each element is assigned a material number which is indexed to parameters which control the constitutive behavior of the element. The program is coded so that solid elements can be connected to a varying number of nodes (between three and eight). The connectivity data for a solid element requires that eight nodes be input for each element. In cases where the element has fewer than eight nodes, the "missing" node numbers are occupied by zeroes. However, for most applications either four or eight node elements are used. This variable node feature can be used to make transitions between elements having dissimilar node configurations in the finite element mesh. Figure 4 shows an example of a 5-node element used as a transition between a 4-node and an 8-node element.

The number of degrees of freedom possessed by the element depends on (a) the type of problem being solved (i.e. the state of FLOW and DEFORM), and (b) the number of nodes connected to the element. For an 8-noded element up to 20 degrees of freedom are possible if a coupled solution is desired (FLOW = DEFORM = TRUE). For such an element, there are 16 displacement degrees of freedom because there are two displacement degrees of freedom for each of the eight nodes connected to the element. The displacement degrees of freedom are activated whenever DEFORM is TRUE. An additional degree of freedom representing the pore pressure is assigned to each of the four corner nodes. The pore pressure degree of freedom is activated whenever the logical variable FLOW is set to TRUE in the data file.

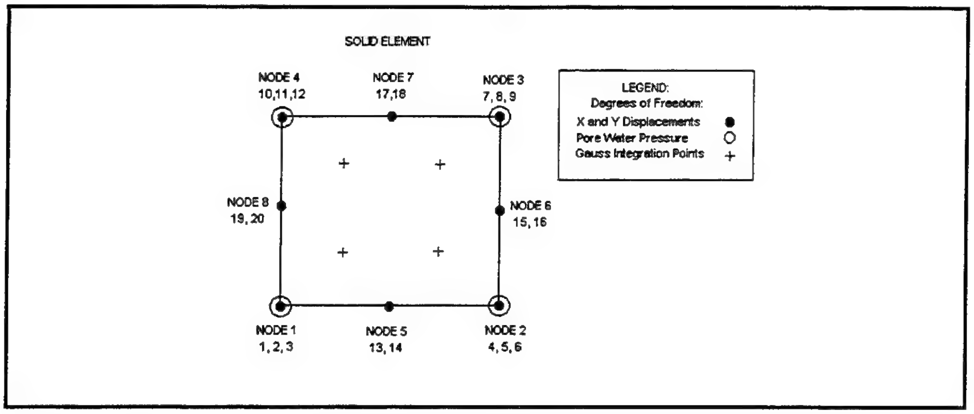


Figure 3. Solid element

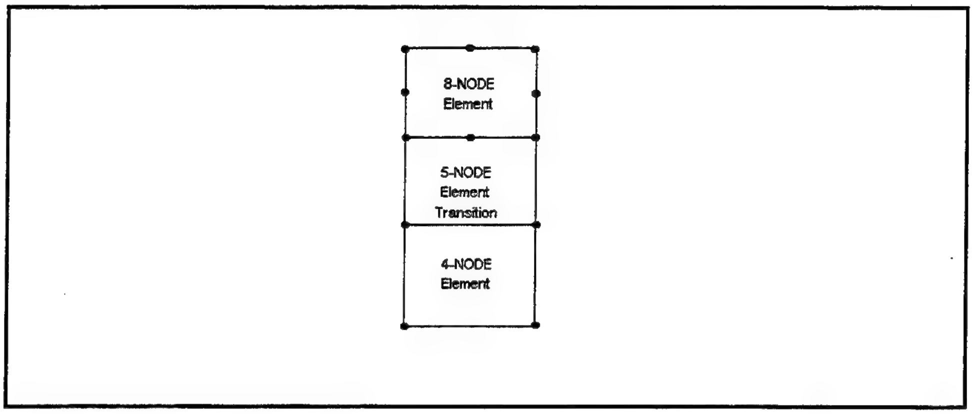


Figure 4. Transition element

The basic rules for building the connection table of a solid element are relatively straightforward. One line in the data file is reserved for each element. Eight positions are reserved for node numbers connected to that element. The first four positions are for the numbers of the corner nodes and the last four positions are for the midside nodes. The corner nodes are listed in counterclockwise order beginning with any of the corners. The midside nodes are then listed beginning with the first midside node on the counterclockwise side of the first corner node.

If a midside node on a particular edge of an element is not present, a zero should be entered into the appropriate midside position in the data file.

Figure 5 represents a series of examples for solid elements connected to various numbers of nodes. The figure illustrates the connectivity for six types of elements. These include: (a) three-node triangular, (b) six-node triangular, (c) four-node quadrilateral, (d) five-node quadrilateral, (e) six-node quadrilateral, and (f) eight-node quadrilateral elements.

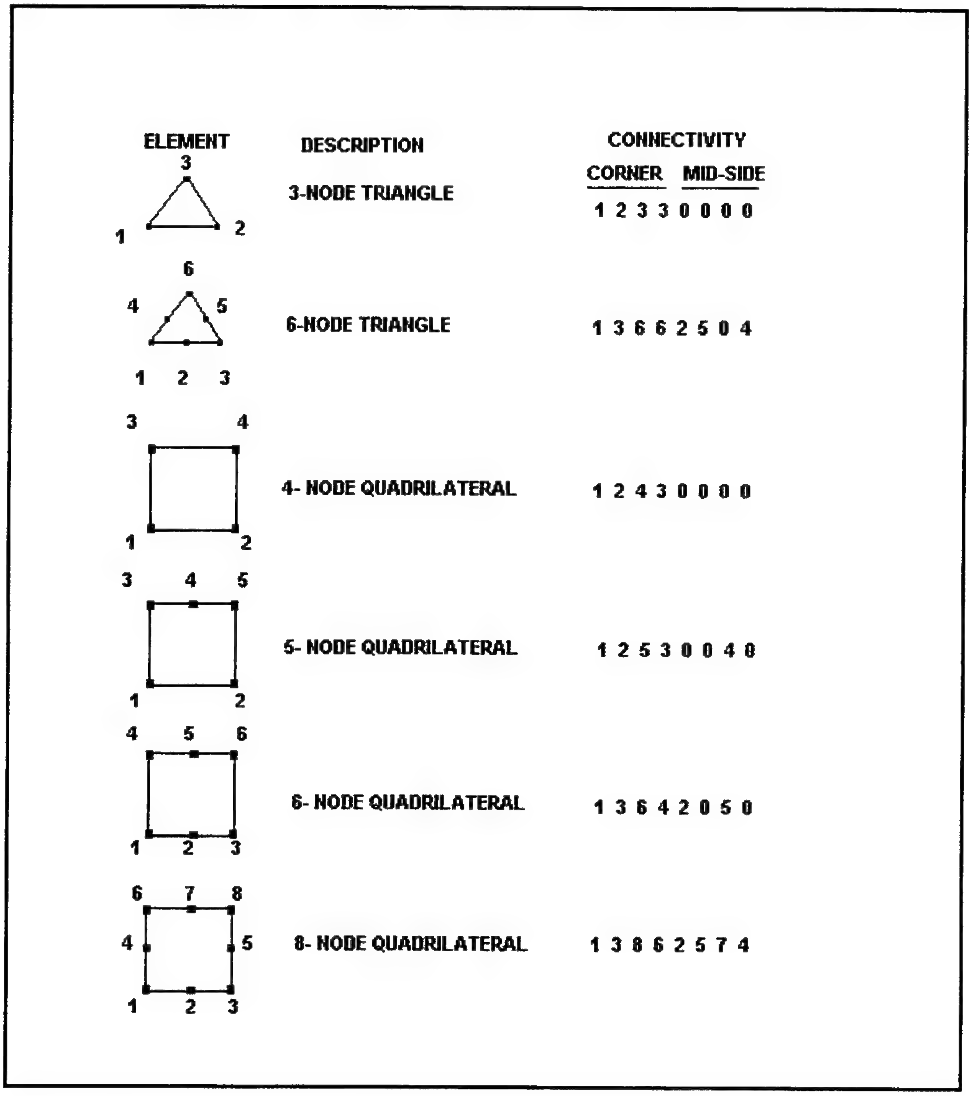


Figure 5. Solid elements

Bar Element

The one-dimensional bar element is used to simulate the behavior of structural components capable of taking either tensile or compressive stresses, or both. The bar is typically used to represent springs, soil reinforcing elements, struts (bracing), and anchors in geotechnical problems. The bar element is capable of resisting loads only in the axial direction. It offers no resistance for load components oriented transverse to the axial direction of the bar.

A schematic representation of the bar element is shown in Figure 6. Essentially, this bar has zero thickness. This figure shows that the bar element can be connected to up to three nodes, two end nodes and one in the

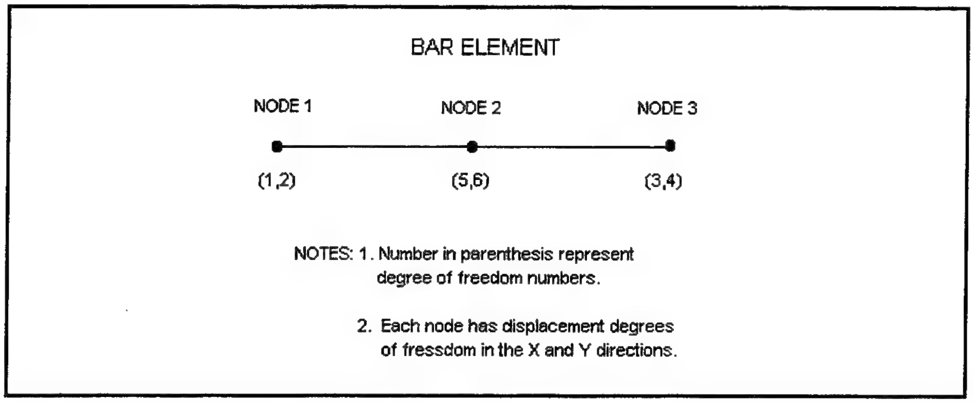


Figure 6. Bar element

center. If two nodes are used, the midside node is left out of the connectivity table. The bar element is only used in coupled or pure deformation type problems because it only permits displacement degrees of freedom (x and y displacements). In coupled problems the bar acts as if it is infinitely permeable and does not inhibit flow.

The bar element is nonlinear in that it may have different tensile and compressive stiffness as shown in Figure 7. Such a material may be termed bilinear. A material which can only take tension should be assigned a tensile stiffness equal to that of the material and a compressive modulus equal to zero. Conversely, a material capable of only taking compressive loads should be assigned the appropriate compressive stiffness and a tensile modulus of zero. The compressive and tensile stiffnesses are equal for linear materials. The stiffness entered in the data file should be represented as the stiffness per unit of width. It is equal to the product of the cross-sectional area per unit of width multiplied by the Young's modulus of the material. As is the case with the solid element, the bar element is assigned a material number which is indexed to the compressive and tensile stiffness. The stiffness of the bar element should not be thought of the modulus of the material. The bar stiffness is equal to the product of the cross sectional area (per unit of width) times the modulus of the material.

Figures 8a, 8b, and 8c show how the bar element can be used to represent soil reinforcement, anchors, and struts, respectively. In the case of the soil reinforcement, the reinforcement is compatible with the solid elements on both sides of the bars because all nodes connected to the bar elements (representing the reinforcement) are also connected to the adjacent soil elements. In contrast, for the case of anchors, the anchors are compatible with the soils only at the endpoints where the nodes are also connected to the appropriate soil elements. Bar elements will always have zero strain when introduced to a problem even though the nodes connected to the bar elements may have some displacements.

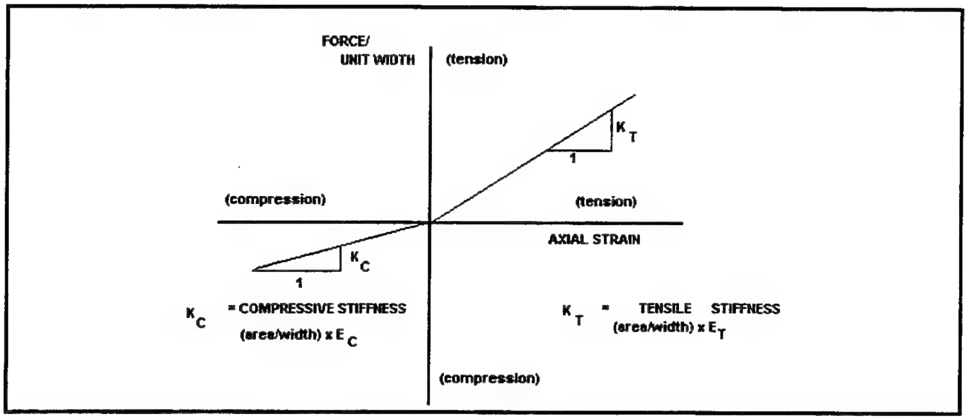


Figure 7. Bar stiffnesses

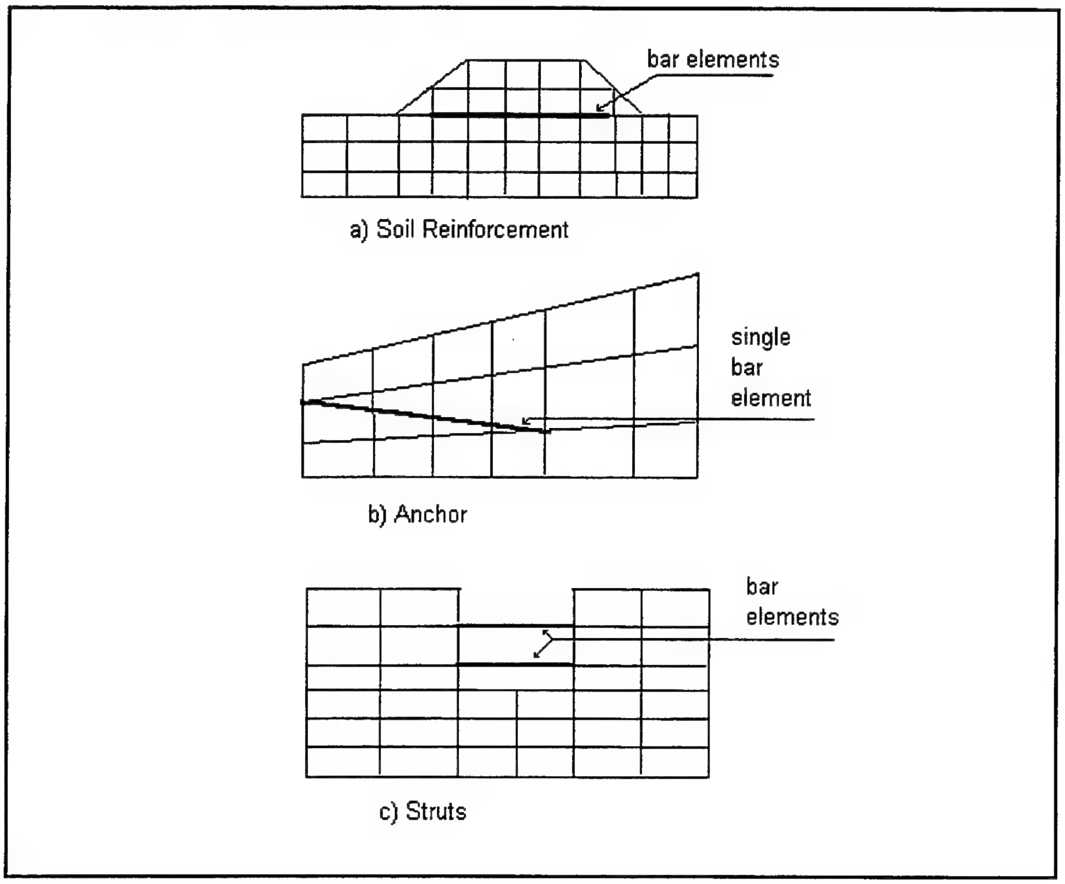


Figure 8. Bar applications

Beam Element

The beam element can resist bending and axial loads. The cross-sectional geometrical properties of the beam elements are not determined by the coordinates of the nodes. Rather, the section properties of the elements are specified in the material properties. The beam element is composed of material that is characterized by an elastic modulus, Poisson's ratio, moment of inertia, and a cross sectional area. The beam element can have either four or six nodes. The six-node element is preferred. The number of nodes on the beam element must correspond to the number of nodes on the element to which the beam element is attached. That is, a beam element attached to a solid element which does not have a mid-side node must, likewise, have no mid-side node. A beam element attached to a element with mid-side nodes must have mid-side nodes.

A typical six-node beam element is shown in Figure 9. The beam elements are compatible with solid, bar, and slip elements. To achieve compatibility, the beam element has the X- and Y- degrees of freedom, but does not have a specific degree of freedom for rotation. Despite the lack of a rotational degree of freedom, the beam element works correctly as a flexural member. In effect, the rotational degree of freedom is replaced by having two displacement values at the beam end. For internal computational purposes, the bending element can be thought of as being composed of three parallel bar elements attached to a solid element. The two outer bar elements resist bending and all three bar elements resist axial loads. The stiffness of the outer bars is determined using the section modulus indirectly input as material properties. The stiffness of the neutral axis bar is determined using the material modulus. The shear stiffness is supplied by the solid element. The initial stresses in the beam elements can be input by specifying the extreme fiber stress in each of the two outer bars. The nodes coded into the connectivity data are treated in the calculations as if the data were two three-node bar elements. A third fictitious bar element is configured using the average position of the two exterior bar elements.

The beam element is anisotropic and intended to simulate structural elements that resist load primarily by bending, such as sheet piling or thin concrete slabs. A thick or tapered retaining wall should be simulated with solid elements rather than with beam elements.

The beam element constitutive relations are based on elastic theory (generalized Hook's Law) rather than beam theory. The use of elastic theory makes the element compatible with solid elements, and soil structure interaction problems converge correctly. The beam element behaves like a deep beam, so that the deflection due to shearing stresses must be considered when comparing the deflection determined by the program to the closed-form solution from elementary bending theory. The deflection due to shearing stresses decrease with the product of the modulus of rigidity and the cross-sectional area.

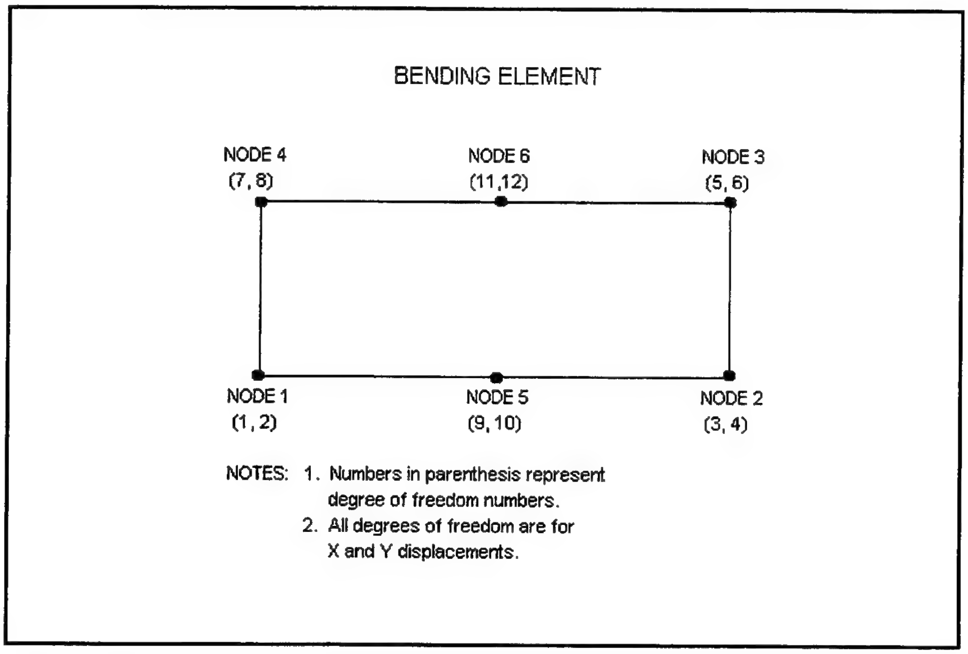


Figure 9. Beam element

Slip-separation Element

The slip-separation element can be a six node or a four node element that has no thickness. The slip element can be used to represent an interface where sliding or separation between two material surfaces may occur. The slip-separation element permits simulation of relative motion between two materials without violating compatibility. The slip-separation element is sometimes called an interface element because of its ability to extend the applicability of the finite element method to model soil structure interaction. The number of nodes on the slip element must correspond to the number of nodes on the element to which the slip element is attached. That is, a slip element attached to a solid element which does not have a midside node must, likewise, have no midside node. A slip element attached to a element with mid-side nodes must have midside nodes. The slip element material properties include strength and deformation parameters. The strength parameters are the friction coefficient, cohesive strength, and tensile strength. The deformation parameters include normal and shear compliance. Compliance is the reciprocal of stiffness.

4 Program Capabilities

The STUBBS program offers two significant capabilities. First, the program can solve the system of equations that describe fully coupled flow and deformation. Second, the program is specifically structured to permit the analysis of construction activities. The program is capable of handling both of the capabilities simultaneously. In addition, the program provides for a variety of finite element types that permit simulation of a soil-structure interaction, and soil and rock reinforcement.

In this part, the user is shown how to access these program capabilities and guidance is provided on setting the initial conditions in the problem. The setting of the initial conditions for coupled flow and partially saturated flow requires careful consideration of computational aspects (how the program works) as well as purely geotechnical considerations.

Time Dependent Problems

Many loading conditions produce a time dependent response in soil and rock. Clearly, time varying loadings produce a time dependent response, but constant loading can produce a time dependent response when flow is required to achieve equilibrium. Not all flow problems produce time dependent responses. For example, steady state flow problems are not time dependent. STUBBS uses a time increment and substeps within a time increment to allow the time dependent response to come to equilibrium.

The keyword time, KW TIME, specifies the length of the time increment for the time step and the number of substeps in the time increment. The KW 'TIME' is required when modeling all time dependent processes such as consolidation.

KW TIME contains two parameters on one line following the KW:

NDT, DT

where

NDT = Number of substeps for this time step

DT = Overall duration of this time increment

Time steps may contain other construction script cards. Time steps are always concluded with the keyword DONE. The KW DONE does not use parameters.

The time step has two effects. First, any loadings applied during the time step are incremented by the number of substeps. For example, if a 10 psi loading was to be applied to a boundary during the time step and the number of substeps was 2, then the loading would be applied in two increments of 5 psi each.

Second, time dependent processes, such as flow, are calculated incrementally in accordance with the time specified in the second parameter of KW TIME.

Construction Simulation

STUBBS uses a script to simulate changes due to construction. The script is a series of keywords that describes the changes in mesh, materials, or boundary conditions. The script must begin with KW LFTC and end with KW DONE.

Construction Activities	Sample KW
Adding and removing material (elements	CSFP, CSEX
Changing material within an element	CSEC
Changing gravity to simulate centrifuge testing	CSNG
Changing boundary conditions	CSBC
Changing loadings	CSPL, CSBL
Changing time increment and number of substeps	TIME

Note that the sample KW's are only a few of the possible KW's that can be used. See Appendix A for a complete listing of KW's.

The user should take a forward-looking point of view when creating the mesh for a construction simulation problem. The user should create the mesh using the maximum dimensions that may exist at any time in the problem. NO NEW ELEMENTS CAN BE CREATED DURING THE PROGRAM RUN. Think of the elements as members of a sports team. During the game different players can be used at the will of the user. The number of players in the game can change from time step to time step. However no new players can be added to the roster. This point of view allows the program to solve for the solution in a very efficient manner.

Initialization

At the beginning of the calculations the initial conditions must be in equilibrium. That is, the pore water pressure, effective stress, flow velocities, and displacements must be compatible. STUBBS attempts to balance the initial conditions set by the user with the internal energy balances that determine equilibrium of each element. In pure deformation problems, pore pressure is not considered and initialization is relatively simple. The user must avoid imposing conditions at the beginning of the problem where static equilibrium is violated.

In flow in partially saturated soil and in coupled flow problems, the pore water pressure, effective stress, flow velocities, and displacements must be compatible. It is sometimes not a simple matter to provide initial conditions that are in equilibrium. If the conditions are not in equilibrium, the calculations may lead to misleading results because the system will still be trying to equilibrate while the problem at hand is being solved. In addition to initialization considerations, the program also automatically adjusts PWP DOF (see PWP DOF, next page) when solid elements are added (fill), removed (excavated), or when an element drops below the water table or the water table drops below a previously saturated element. An understanding of the program conventions may assist the user in controlling these automatic actions to get the correct problem formulation.

Automatic degree of freedom (DOF) actions

The displacement and pore water pressure degrees of freedom can be explicitly controlled by Keywords. The STUBBS program automatically alters some of the DOF's depending on the nature of the problem and the degree of saturation of the element. Among the automatic actions are:

a. Displacement DOF are turned off for all nodes in pure flow problems.

b. PWP (pore water pressure) DOF are turned off for all nodes in pure deformation problems.

c. PWP DOF are turned off for surface nodes in coupled problems.

d. PWP DOF are turned on for nodes on a permanent boundary.

e. PWP DOF are turned off when an element reaches the Residual Saturation (RSAT) for the material in pure flow and coupled flow.

f. PWP DOF are turned on when an element should return to saturation greater than the RSAT in pure flow and coupled flow.

Of these automatic procedures, the practice of turning off the PWP DOF for surface nodes must sometimes be explicitly overruled by using the KW CSBC. For example, if the free (water table) surface is above the ground

surface, the user must turn on the PWP DOF in the nodes on the ground surface. The practice of turning off the PWP DOF for surface elements allows fill placement or excavation to proceed without the user manually turning off the PWP DOF for new elements. Nodes are assigned a status depending on their situation at the start of a time step. A brief discussion of status labels and the status reports follows.

Status reports

Two classes of information are presented as status reports. One class is initial data that includes an echo of the data drawn from the input file plus a table presenting the suction-saturation relation for the soils (solid materials). The other class of information is a status of elements at the beginning of each time step. The element numbers shown in the summary table are the global numbers used within the calculation subroutines of STUBBS. The local (element type) numbers are used in all other tables. For example, a beam element input by the user as beam element 1 may be global element 14. The solid elements are assigned global numbers starting at 1, and then bars, slips and beams are assigned numbers in that order. The user may wish to know the global number assigned within the program. The summary table is the only place where the global element numbers can be verified.

The initial data are always sent to the output file. The KW VBOF turns off the status report within each time step. The status report can be turned on by using KW VBON. The default is VBON.

The headings for the initial data are:

Problem Summary
Nodal Point Data
Soil Element Connectivity Data
Initial Values in Solid Elements
Material Property Data
Pressure-Saturation Curve

The headings for the time step data are:

Summary of Loading and Construction Events for Step n
Summary of Activities
Summary of Active Elements
Summary of Active Nodes

The possible status labels for elements in the "Summary of Active Nodes" are surface boundary (SB), permanent boundary (PB), excavated boundary (EB), and interior (IN). The definitions of these terms are shown graphically in Figure 10 and in shown in Table 6 .

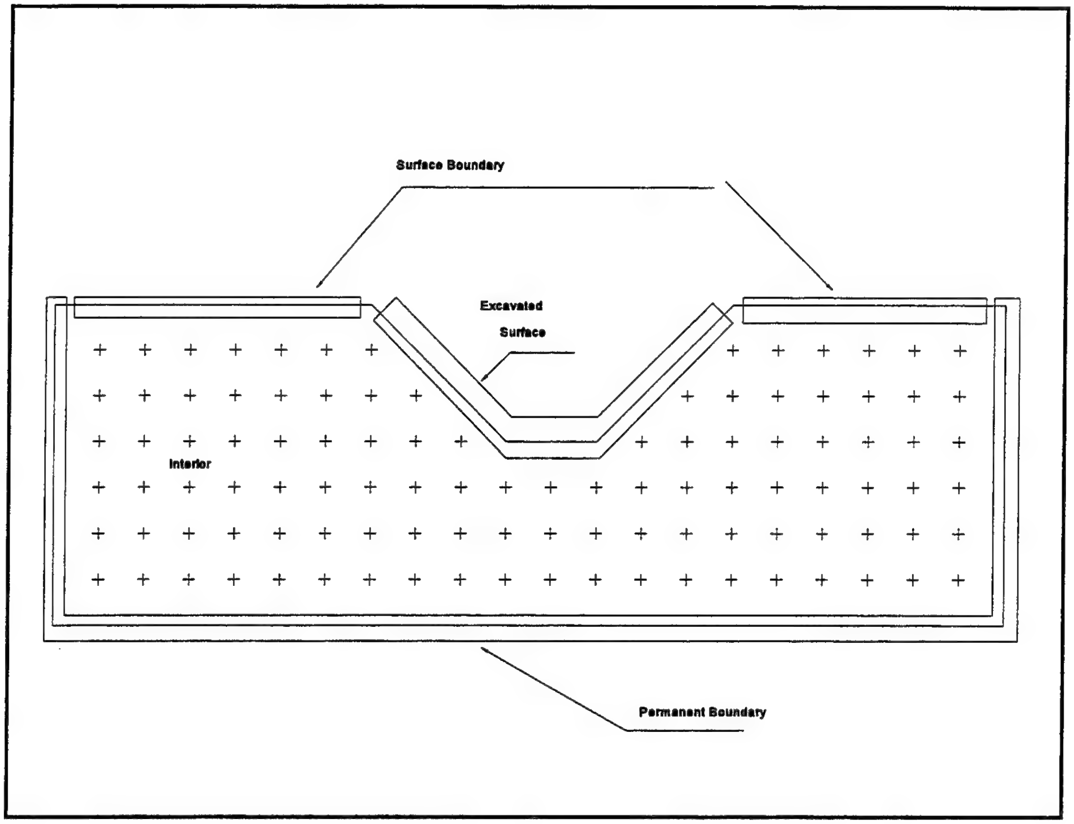


Figure 10. Node status labels

Table 6 Definitions Used in Summary of Active Nodes	
Surface boundary (SB)	The node bounds a side that is part of a free surface and does not have a displacement DOF fixed. This status is established as an initial condition. If the node forms part of a excavated surface the status of the node becomes PB, but the former SB node remains fixed unless freed by KW CSBC.
Permanent boundary (PB)	The node bounds a side that is not shared by another element at the start of a time step and has one or both displacement DOF's fixed, or is part of an excavated surface.
Excavated boundary (EB)	The node bounds a side that was shared by an element removed in the current time step. EB nodes are reclassified to PB nodes in subsequent time steps. During the first time step as a PB node, the PWP DOF is fixed and then turned free in remaining time steps.
Interior (IN)	The node bounds a side that is shared by two elements.

The following rules apply to PWP DOF program conventions:

a. The pore pressure degrees of freedom on SB nodes are always fixed, unless freed by KW CSBC. KW NODE and KW IDPP are overruled. When freed by KW CSBC, the status of the node becomes PB.

b. The pore pressure degrees of freedom on PB nodes are fixed for the time step when the status is established, and free for subsequent steps, unless the node was initially SB. SB nodes remain fixed unless freed by KW CSBC.

c. The pore pressure degrees of freedom of IN nodes are free, unless fixed by KW CSBC or KW NODE.

Preferred practice

In flow and coupled problems it may be necessary to carry out separate calculations to establish the initial conditions for the problem at hand. In this calculation whose goal is to establish the "true" initial conditions, the user can apply the appropriate boundary conditions to the mesh and arbitrary pore water pressures to the nodes. The user then scripts a long stabilizing period to bring the system to the steady state. The pore water pressures and stresses from the last time step will represent the steady state. The quantities should then be input into the data file for the problem at hand under KW IDPP and as they represent the "true" initial conditions for the problem at hand. Examples of initialization for flow and coupled flow problems are shown in Appendix C.

5 Summary

General

The STUBBS program is a script-driven finite element program. The program is a 2-D plane strain program that can also model axisymmetric conditions. The program was specifically designed for geotechnical analysis. The program solves deformation problems, flow problems, and coupled flow/deformation problems. The program is capable of modeling construction processes such as excavation and filling through the use of a construction script. The script is built up of keywords. The general operation of the program was presented in Chapter 1 Description of STUBBS. Additional information on the keywords is provided in Appendix A: Keyword Dictionary. The dictionary provides a complete list of keywords and illustrates the syntax of each keyword used in an example.

The keyword script is placed within a data file. The data file and a control file called FEA_FILE must be prepared by the user. The details regarding preparation of input files is provided in Appendix B: File Structure. Additional guidance on data files is presented in the example problems in Appendix C.

The program can use a variety of element types to represent soil, rock, concrete, geogrid, sheet piling, and anchors. The description of the element types was presented in Chapter 1. The solid element is the principal element used to model soil and rock. The solid element is the only one of the four element types that is capable of modeling flow. A detailed description of the solid elements was provided in Chapter 2 Material Behavior of Solid Elements.

The additional elements, beam, bar, and slip-separation, were created to model retaining structures and soil and rock reinforcement. The capabilities of these elements was described in Chapter 3 Finite Element Library.

A set of example problems is presented in Appendix C. The problems illustrate the use of each type of element. The problems are indexed by element type and by keyword. An alphabetical listing of examples is also provided.

User Comments

Comments and suggestions regarding this manual are welcomed. Users should address the comments to Dr. John Peters, U.S. Army Engineer Waterways Experiment Station (CEWES-GS-GC), 3909 Halls Ferry Road, Vicksburg, MS 39180-6199.

References

Biot, M. A. (1941). "General theory of three-dimensional consolidation," *Journal of Applied Physics*, Vol 12, pp 155-164.

Peters, J. F. and Valanis, K. C. (1993). "Experiences with numerical implementation of endochronic models," *Modern Approaches to Plasticity*, D. Kolymbas (ed.), Elsevier Science Publishers, B. V., Amsterdam, The Netherlands.

Valanis, K. C. (1980). "Fundamental consequences of a new intrinsic time measure plasticity as a limit of endochronic theory," *Archives of Mechanics*, Vol 32, pp 171-191.

van Genuchten, M Th. (1980). "A closed-form equation for predicting the hydraulic conductivity of unsaturated soils," *Soil Sci. Soc. Am. J.*, Vol 44, pp 892-898.

Valanis, K. C. and Peters, J. F. (1991). "An endochronic plasticity theory with shear-volume coupling," *Int. Journal for Numerical and Analytical Methods in Geomechanics*, Vol 15, pp. 77-102.

"Geotechnical analysis by the finite element method." (1995). Technical Letter No. 1110-2-544, Department of the Army, U.S. Army Corps of Engineers, Washington, DC 20314-1000.

Appendix A
Keyword Dictionary

The following keywords are used in the computer program STUBBS. These keywords are identified in a data input file by the descriptor 'KW.' Following the alphabetical list is a syntax guide organized by keyword function.

The following is an alphabetical listing of the 42 keywords.

BEND - Lists beam elements connectivity data (6-nodes).

BMMP - Provides the material properties for beam elements.

BREL - Lists the bar element connectivity data (2 or 3-nodes).

BRMP - Provides the bar element material properties.

CSBC - Indicates that the boundary conditions for a specified list of nodes have changed from the previous time step. For example, node 100 can be changed from having FIXED-FIXED displacement boundary conditions to FREE-FREE boundary conditions using **'CSBC'**.

CSBL - Identifies that distributed loads are to be applied normal to a specified boundary (defined by a list of nodes) during a given time step.

CSBP - Lists bar elements which are to be installed during a given time step.

CSBR - Lists bar elements which are to be deleted during a given time step.

CSBU - Lists elements which are to be submerged or emerged during the current time step.

CSEC - Contains data changing the material type of an element. For example, an element can be excavated during one time step as one material type and filled back in during the current time step as a different type of material.

CSEX - Lists the solid elements which are to be excavated during the current time step.

CSFL - Indicates that a list of beam elements is to be added during the current time step.

CSFP - Indicates that a list of solid elements is to be added during the current time step.

CSFR - Indicates that a list of beam elements is to be removed during the current time step.

CSNG - Changes the acceleration due to gravity. This feature is used to model centrifuge testing.

CSPL - Identifies nodes that are to receive concentrated loads during a given time step.

CSSP - Identifies a list of slip-separation elements to be added during the current time step.

CSSR - Identifies a list of slip-separation elements to be removed during the current time step.

DONE - Signifies that all construction data for the current time step has been completely read in. There must be one **'DONE'** card for every **'TIME'** card. Additionally, there must be one **DONE** card for the **LFTC** card which signifies that the total construction sequence and modeling time period have been completed.

ELEM - Specifies and defines all of the solid (soil) elements of the problem. Element data includes the element number, the material type, and the nodal connectivity (3 to 8 nodes).

ENDO - Activates the endochronic model. Contains material properties for plastic soil behavior that supplement the linear elastic properties provided in **SMPR**.

IBND - Initial state of preexisting bending elements.

IDPP - Initial node data. This keyword is used to specify prescribed values and boundary conditions of preexisting nodes.

ISPR - Provides initial stress and state data for preexisting solid elements.

ISRE - State of preexisting bar elements in terms of stresses.

ISSL - Specified displacements for preexisting slip elements.

LFTC - Marks beginning of the construction sequences and data. KW 'LFTC' is required for all problems which have construction steps.

NODE - Specifies the position and boundary conditions of each of the nodal points for the entire problem. In deformation problems, nodes have x and y degrees of freedom. In flow problems, corner nodes have pore pressure degrees of freedom. In coupled deformation/flow problems, node points have x and y displacement and corner nodes also have pore water pressure (corner nodes) degrees of freedom. Nodal coordinates for x and y are input in the units designated by **KW UNIT** and can be adjusted by x- and y- scale factors.

RPOF - See **RPON**.

RPON - Controls the writing of a report of results (stresses, displacements, pore water pressures) to the output file for a given time step. The **KW RPON** toggles with the **KW RPOF**. When **RPON** is active, the results are written after each time step. When **RPOF** is active, no results are written. The default in STUBBS is **RPON**.

SLIP - Lists slip elements connectivity (4 or 6 nodes)

SLMP - Provides material properties for slip elements

SMPR - Provides the material parameters for solid elements

SPEC - Initializes the logical variables which determine the type of problem which will be solved. The logical variables include deformation, flow, and axisymmetry. Drainage conditions are specified. Also, **SPEC** contains information which specifies whether or not the mesh and construction script are to be checked.

TIME - Indicates a construction time step of a given duration and divided into a number of substeps.

UNIT - Specifies the dimensions in which the problem will be worked out in. The dimensions are defined in terms of the units of atmospheric pressure, the unit weight of water, and acceleration due to gravity. In addition, the user may specify a convergence criterion, maximum number of iterations and time integration constant. If the user does not specify any criterion, default values of 1E-8 units of force for the residual, 8 iterations, and a time integration constant of 0.6667.

VBOF - See **VBON**.

VBON - Controls the writing of a report containing the status of elements and nodes to the output file. The **KW VBON** toggles with the **KW VBOF**. When **VBOF** is active, a partial report is written to the output file, when **VBON** is active, a full report is written to the output file. The default in STUBBS is **VBON.**

XBND - Identifies preexisting beam elements

XBRE - Identifies preexisting bar elements

XELT - Identifies elements which exist at the start of the problem. Preexisting elements are input as a list or, if sequential, they can be input over a specified range of elements separated by a colon

XSLP - Identifies preexisting slip elements.

Syntax and Usage of Key Words (KW)

Initialization group keywords

UNIT

INPUT:
KW UNIT
(one line) Patm, GAMMAW, GRAVITY, CRESID, NPASS, THETA
Patm = atmospheric pressure (F/L^2)
GAMMAW = unit weight of water (F/L^3)
GRAVITY = acceleration due to gravity (g's)
CRESID = convergence tolerance (unit less) [default value = 1E-8]
NPASS = maximum number of iterations [default value = 8]
THETA = time integration constant (dimensionless)[default value = 0.6667

Notes:

a. This keyword specifies the units which will be used in the finite element solution. The three units parameters must be provided.

b. All units for PATM and GAMMAW must be selected so that they are consistent in terms of force (F), length (L), and, indirectly, time (T). NOTE that the units of gravity are in g's. The units used for time must be compatible with the material properties for solids using the KW SMPR or ENDO.

c. Only one line is required. The convergence criterion, the number of iterations, and the time integration constant are optional.

d. CRESID is the convergence criterion which is compared with the residual computed internally by STUBBS. The residual is a normalized error and is defined as the square root of the sum of the square of the differences between the applied force and the equivalent nodal forces from the internal stresses divided by the square root of the sum of the square of the applied forces. The iterations during a given time step involving a nonlinear problem will stop when the residual is less than the CRESID or when the number of iterations equals NPASS.

Example:

KW UNIT
C Patm, Gammaw, Gravity (3 required fields, 3 optional fields)
C convergence criteria, max number of iterations, time integration constant
14.7 0.0361 1.0 1E-6 12 0.8

In this example, force is lbs, length is inches, the solution will be considered converged if the difference between the applied and calculated force is less than 1E-6. The number of iterations available to reach convergence is 12. The time integration constant is 0.8.

SPEC

INPUT:

KW SPEC
C FLOW, DFORM, DRAIN, CHECK_MESH, CHECK_SCRIPT, AXIS
(6 fields)

FLOW = Variable which indicates whether or not flow equations are to be included in the solution.
DFORM = Variable which indicates whether or not deformations are to be included in the solution.
DRAIN = Variable which indicates whether or not drained conditions are to be modeled in the solution.

CHECK_MESH = Variable whose state will determine whether or not the data input file will be checked for errors. If CHECK_MESH is set to TRUE, then the input data will be checked for its correction and the calculations will not be begun. If CHECK_MESH is set to FALSE, then the finite element problem will be solved after the data file is read in, provided that the data file contains no errors.

NOTES:

a. SPEC initializes the logical variables which determine the nature of the problem to be solved.

b. All variables are logical and are input as either TRUE or FALSE.

Example:

KW SPEC
C FLOW, DFORM, DRAIN, CHECK_MESH, CHECK_SCRIPT, AXIS
(6 fields)
F T T F F F

In the example, checking the mesh and script features are turned off. The last "F" indicates that axisymmetry is turned off, so this problem is a plane

strain problem. Flow is turned off and deform and drain are turned on. This set up is suitable for a problem involving pure deformation.

Hint: If either CHECK_MESH or CHECK_SCRIPT is set to T no solution is attempted after the input data is checked. Use CHECK_MESH and CHECK_SCRIPT to check the completeness of the data and then switch the variable to F when satisfied that the input data is correct and complete.

NODE

INPUT:

KW NODE
(line 1) xscale factor, yscale factor (2 fields)

x-scale factor - real number use to scale (i.e. multiply coordinates) x-coordinates
y-scale factor - real number use to scale (i.e. multiply coordinates) y-coordinates

(lines 2 and higher) number, x-coord, y-coord, xdisp code, ydisp code, pw code (6 fields) number is a integer node name (1, 24, etc.)

x-coord is the value of the x-coordinate of the node on a Cartesian plane. In an axisymmetric problem, this x-coord is the radial or r-coordinate distance from the axis of symmetry, that is the r = 0 line.

y-coord is the value of the y-coordinate on a Cartesian plane. The y-direction is vertical and up-movement is positive. The z-axis is horizontal and there is no strain in the z-direction (plane strain condition). In axisymmetric problems, the z-direction is replaced by the theta direction.

xdisp code - xdisp = 1, fixed; xdisp = 0, not fixed; xdisp > 1; then assigned a prescribed value as per equations in subroutine boundary function.

ydisp code - ydisp = 1, fixed; ydisp = 0, not fixed; ydisp >100; then assigned a prescribed value as per equations in subroutine boundary function.

pw code is the degree of freedom for pore water pressure.
pw code = 1, pore water pressure has a fixed (unchanging) value.
pw code = 0, pore water pressure is free to change, subject to calculation.
pw code > 1, then assigned a prescribed value as per equations in subroutine boundary function.
PW code = 100, pore water boundary condition is initially free as long as the pore water pressure is negative. The boundary condition will change to a fixed condition if the pore water pressures becomes positive during

the calculation. The porewater pressure for the new fixed condition will be set to zero. This feature is useful in determining the free surface on exterior boundaries for seepage problems.

Example:

```
KW NODE
C  x-scale factor   y-scale factor (2 fields)
   1       1
C node # x-coord  y-coord  x-disp  code     y-disp code    pw code  (6 fields)
   1       0        0        1       1          0
   2       1        0        0       1          0
   3       1        1        0       0          0
   4       0        1        0       0          0
```

In the above example, the x-scale factor and y-scale factor are set to 1.0. Four nodes are input forming the corners of a square. Node 1 is fixed in both the x- and y- directions. Node 2 is fixed in the y-direction. Nodes 3 and 4 are free to displace. All of the nodes have pore water degree of freedom set to free, that is, free to change. Note that only corner nodes of solid elements have pore water degrees of freedom with respect to program execution. Mid-side nodes of solid elements and all nodes of bending, bar and slip-separation elements have no pore water degree of freedom. Nevertheless, the field provided for pw code must be filled for every node. In general, because of the reasons mentioned above, this field is a filler and the value shown in the field (1,0, or other number) has no effect on calculations except for the corner nodes of solid elements when the logical FLOW variable in KW SPEC is set to True (T).

Defining Elements

After the nodes have been input using KW NODE, the nodes must be formed into elements as defined by one or more of the following KW's:

ELEM (solid elements)
BEND (bending elements)
BREL (bar elements)
SLIP (slip-separation elements)

ELEM - Configures solid elements

INPUT:

KW ELEM
C element number, material number, c1, c2, c3, c4, m1, m2, m3, m4

where

Element number is an integer with element numbered from 1 to n, sequentially. material number is an integer corresponding to a material type whose properties are defined using the KW SMPR or KW ENDO.

c1, c2, c3, c4 are the node numbers for the four corner nodes entered in counter-clockwise order with respect to the element center.

m1, m2, m3, m4 are the four midside nodes entered in counterclockwise order, starting with the midside node that lays between corner nodes c1 and c2.

Example:

```
KW ELEM
C  element number, material number, c1, c2, c3, c4, m1, m2, m3, m4
1 1  12  15  22  23   2  11 18 26
2 3   6   7   8   8   0  0  0  0
3 1   3   7  10  16  24 27  0  0
```

BEND - Configure bending elements

INPUT:

```
KW BEND
C   element number, material number, c1, c2, c3, c4, m1, m2
```

where

The parameters are defined somewhat similar to the those in KW ELEM except that bending elements must have six nodes, four corner nodes and two midside nodes. If four node beam elements are used, the midside nodes (m1, m2) are listed as zero. The material number must correspond to a material defined using the KW BRMP. Bending elements are numbered from 1 to n, sequentially.

Example:

```
KW BEND
C   element number, material number, c1, c2, c3, c4, m1, m2
        1               2                 4   12    22   6    9  25
        2               1                12   25    30  29   13  18
```

BREL - Configure bar elements

INPUT:

```
KW BREL
```

C element number, material number, end node, opposite end node, midside node (5 fields)

where

Bar elements are numbered from 1 to n, sequentially. Bar element may have either three nodes or two nodes. In a two-node bar, the midside node field is zero. The bar material must be a number defined using KW BRMP.

Example:

```
KW BREL
C element number, material number, end node, opposite end node, mid-
side node
1 4 3 9 6
2 4 5 7 0
```

SLIP - Configures slip elements

INPUT:

KW SLIP
C element number, material number, c1, c2, c3, c4, m1, m2, 0, 0

where

The parameters are defined similar to the those in KW ELEM except that slip elements must have six nodes, four corner nodes and two midside nodes when used with eight-node solid elements. The last two fields for midside nodes are filled with zero. When used with four node solids, the slip elements have four corner nodes and no midside nodes. The material number must correspond to a material defined using the KW SLMP. Slip elements are numbered from 1 to n, sequentially.

Example:

```
KW SLIP
C  element number, material number, c1, c2, c3, c4, m1, m2, 0 ,0
    1         2                  4    12    22   6         9   25  0  0
    2         1                 12    25    30  29        13   18  0  0
```

Preexisting Elements

The element numbers for those elements present at the start of the problem will be defined by one or more of the following KW's:

XBND - Identifies preexisting beam elements
XBRE - Identifies preexisting bar elements
XELT - Identifies solid elements which exist at the start of the problem
XSLP - Identifies preexisting slip elements

The KW's defining existing elements (KW X...) must NOT appear before the KW that form the elements. For example, the XBRE must not appear before BREL or the program will fail to configure the bar elements.

INPUT:

The input for each of these X... KW's is the same. KW XELT will be illustrated.

```
KW  XELT
C  element list
```

The element list can be enumerated as 3 4 5 6, or for sequential numbers, the first and last may be separated with a colon as 3:6.

Example:

```
KW XELT
C element list
1  5  7  12  20:25
```

Material Properties

The elements will be composed of a material type defined by one or more of the following KW's:

SMPR - Solid elements
ENDO - Solid elements
BMMP - Bending elements
BRMP - Bar elements
SLMP - Slip elements

SMPR- Solid Element Type Data

Linear Elastic Materials. The constitutive law for linear elastic materials is Hooke's law. Among the material parameters required for linear elastic solid elements are Young's Modulus, Poisson's Ratio, the at-rest earth pressure coefficient, permeabilities in the X- and Y- directions, and two parameters describing the soil suction-permeability relation. The two soil suction parameters are called van Genuchten parameters and are discussed in detail in a following subsection. In addition, anisotropy may be considered by providing different flow and compressibility parameters for horizontal and vertical

directions. Both permeability and elasticity properties correspond to a transversely isotropic material. In the present version, the principal material axes coincide with the x, y and z directions.

This keyword is used to input the material properties of solid elements.

INPUT:

KW SMPR

Line No. 1: M, GAMMAT, RVOID, GS, RSAT (5 fields)
Line No. 2: E2, P1 ,P2, XN, XM , Ko (6 fields)
Line No. 3: PERM1,PERM2, SKEMPTON_B (3 fields)
Line No. 4: Alpha, Beta (2 fields)

where:

M = Material number (integer)
GAMMAT = Unit weight of soil (same units as GAMMAW)
RVOID = Void ratio
GS = Specific gravity of solids
RSAT = Residual degree of saturation (residual saturation, see van Genutchen parameters discussion.)
E2 = Young's modulus (same units as Patm)
P1 = Poisson's ratio (X-direction)
P2 = Poisson's ratio (Y-direction)
XN = Ratio of horizontal stiffness to vertical stiffness
XM = Ratio of Shear stiffness to vertical stiffness
Ko = Earth pressure coefficient
PERM1 = Coefficient of permeability in X-direction using units of length consistent with Patm, but the units of time are governed by this variable
PERM2 = Coefficient of permeability in Y-direction using same units as PERM1
SKEMPTON_B = Skempton's B parameter (must be greater 0)
Alpha = van Genuchten parameter (using units of 1/Patm) (see Part II)
Beta = van Genuchten parameter (see Part II)

Example:

```
KW SMPR
C  M, GAMMAT, RVOID, GS, RSAT  (5 fields)
          1  0.069        0.8   2.7     0.85
C  E2, P1 ,P2, XN, XM, Ko (6 fields)
         1.5E5    0.35  0.35  1.0 1.0    0.5
C  PERM1,PERM2, SKEMPTON_B  (3 fields)
          1E-4  1E-4        0.99
C  Alpha, Beta (2 fields)
        0.0083           2.3
```

ENDO

Nonlinear Materials. Nonlinear material behavior is represented in STUBBS by the endochronic model (Valanis and Peters 1992). The endochronic parameters are not described in this report. A report on the enchronic parameters is in preparation. This example is provided for completeness of the dictionary only.

The user must use KW SMPR *in addition to* KW ENDO. The nonlinear behavior of this model can be visualized as a system of springs and dashpots. The endochronic model offers the advantage of unloading without any special parameters and has the capability of modeling cyclic loading exceptionally well. Fourteen parameters (plus the material number) are required to describe the stress-strain characteristics of endochronic materials. For a particular soil, these parameters can be determined from analysis of laboratory tests. A procedure is currently being developed which will allow the parameters to be determined from knowledge of conventional soil mechanics properties such as friction angle, cohesion, coefficient of consolidation, and initial void ratio. The endochronic material assumes the elastic component of response is isotropic.

Example, (this example is provided for completeness in describing KW's.)

```
KW ENDO   ....Soil properties for plastic response.
C ===================== Foundation
Parameters=================
C ======== Clay 1 Su = 250 psf 82% level =========
C M    Fo_H    BETA    Fo_S    A_11
  1    1.E05   6.3     335.0   0.000001
C
C Mcv      A_22     Co
  1.1      0.001    0.001
C
C A1     A2      A3      A4
  0.03   0.740   0.220   0.01
C
C  ALPHA2     ALPHA3     ALPHA4
   3.25E6     1.0E5      100.0
```

BMMP

Beam Element Material Type

INPUT:

```
KW BMMP
     M, E, POIS, I_SEC, A_SEC, WEIGHT
```

where

M	=	Beam material number
E	=	Young's Modulus (units of Patm)
POIS	=	Poisson's ratio
I_SEC	=	Cross-sectional moment of inertia (units consistent with Patm)
A_SEC	=	Cross-sectional area (units consistent with Patm)
WEIGHT	=	Unit weight of material (units of Gammaw)

Example:

```
KW BMMP
C  M,     E,      Pois,    I,     A,     unit weight (6 fields)
   1      3E6     0.25     1000   2      45
```

BRMP

Bar Element Material Type

This keyword identifies the material properties for bar elements. One bar material for each different bar material in mesh.

INPUT:

```
KW BRMP
M, AEtensile, AEcompressive
```

where

M	=	Bar element material number
AE (tension)	=	Bar element stiffness in tension (Area*Young's Modulus;AE) using units consistent with Patm. Also, the stiffness should be adjusted to be appropriate for a unit thickness in a plane strain problem. The nature of the adjustment is problem specific.
AE (compression)	=	Bar element stiffness in compression with the same units as AEtensile

Example:

```
KW BRMP
C  Mat #,  AE (tension), AE (compression)
   1    1E-12   3E6
```

In the above example the tensile stiffness was made very low and the compression stiffness is substantial. This material is stiff in compression, but has no tensile resistance.

SLMP - Slip-separation Element- Interface Material Type

Keyword identifying the interface material properties for slip-separation elements.

INPUT:

KW SLMP
C Mat #, tension, friction, cohesion, normal compliance, shear compliance

where

Mat #	=	slip-separation element material number
tension	=	strength in pure tension (units of Patm)
friction	=	friction coefficient
cohesion	=	no-load shear strength (units of Patm)
normal compliance	=	interface deformability (units of 1/Patm)
shear compliance	=	interface deformability (units of 1/Patm)

Example:

```
KW SLMP
C Mat#, tension, friction, cohesion, normal compliance, shear compliance
   3     12.0     0.5     100     1E-5     1E-8
```

Initial Conditions

The KW's beginning with "I" indicate the stresses or displacements in elements that should be applied at the start of the calculations.

IBND - Initial state of stress of preexisting bending elements.

INPUT:

KW IBND - Initial state of stress of preexisting bending elements.
C beam element number, initial stress in extreme top (or left) fiber, initial stress in
C bottom (or right) fiber (3 fields)

Example:

```
KW IBND
C  element #, top fiber stress, bottom fiber stress
  2    0.0      0.0
  5  120.0    120.0
```

8 100.0 80.0

The bending element must have been declared preexisting by use of the KW XBND.

IDPP - Initial node data, displacement and pore pressures

INPUT:

KW IDPP
C node #, X displacement, Y displacement, pore pressure (3 fields)

The displacements are consistent with the units used implied by Patm. The pore pressure units are identical to those used for Patm. CAUTION: IDPP requires that all nodes be listed in ascending order. If a node is omitted or placed out of order, unpredictable values will be assigned to the nodes. IDPP is not required but is recommended for flow and coupled flow problems.

Example:

KW IDPP
C node #, X displacement, Y displacement, pore pressure (3 fields)

1	0.0	0.0	0.0
2	0.0	-0.12	3.0
3	-2.0	-0.9	0.0

ISPR - Provides initial stresses and state data for preexisting solid elements.

INPUT:

KW ISPR
C element number, sigma xx, sigma yy, sigma zz, tau, void, sat
(7 fields)

where

the element number must be a preexisting element as defined by KW XELT
sigma xx is the normal stress acting on a vertical Y-Z plane (units of Patm)
sigma yy is the normal stress acting on a horizontal X-Z plane (units of Patm)
sigma zz is the normal stress acting on vertical X-Z plane (units of Patm)
tau is the shear stress acting on a horizontal plane (units of Patm) void is the void ratio
sat is the degree of saturation

Example:

```
KW ISPR
C  element number, sigma xx, sigma yy, sigma zz, tau, void, sat
(6 fields)
                    2       0.0    0.0   0.0   0.0    1.0   0.9
                    4       1.89 -1.0  -0.3  -0.77  0.8   1.0
```

Note that element 2 has initial stresses of zero. Element 4 has a sigma xx of 1.89 (units of Patm), a sigma yy of -1, a sigma zz of -0.3, and a shear stress of -0.77. The horizontal and vertical planes are not principal planes in element 4. The stresses are applied to the center of the element.

The solid element must have been declared preexisting by use of the KW XELT.

ISRE - State of preexisting bar elements in terms of stresses.

INPUT:

```
KW ISRE
C  bar element number, stress, stress
```

Example:

```
KW ISRE
C  bar element number, stress, stress
1                0.0              0.0
2               10.0              0.0
```

The bar element must have been declared preexisting by use of the KW XBRE.

ISSL - Specified displacements for preexisting slip element nodes.

INPUT:

```
KW ISSL
C  slip element node number, X-displacement, Y-displacement
```

Example:

```
KW ISSL
C  slip element node number, X-displacement, Y-displacement
1      0.0     0.0
2     -0.33    0.0
```

The slip element must have been declared preexisting by use of the KW XSLP.

Construction Script

KW's beginning with "C" indicate a change in conditions due to construction. The construction script ALWAYS begins with the KW LFTC and is concluded with the KW DONE. These two KW's, LFTC and DONE, each stands alone on a line and have no additional parameters associated with them. Within a script, the construction events occur during a time step. The KW TIME serves to indicate the start of a time step or construction event and increments the construction changes into subevents. The KW TIME has two parameters associated with the KW; the number of substeps or subevents and the actual duration of the entire event. The event is terminated with the KW DONE. Each KW TIME together with additional construction KW's and the terminating KW DONE is called a time step.

TIME

INPUT:

```
KW TIME
C   NDT    DURATION
```

where

NDT = the number of substeps or subevents (an integer)
DURATION = actual elapsed time for an event to occur (units of time consistant with the units used to specify permeability)

Example:

```
KW TIME
C    NDT   DURATION
           5      1     .
. (additional construction script KW's)
.
KW DONE
```

The duration is 1 second, minute, day, month, or year, consistent with the unit of time measure used in assigning permeability with KW SMPR. In linear elastic problems without flow, the unit of time is not a factor in the solution. The integer NDT divides the solution into a number of substeps. In problems where loads or boundary pressures are changes the number of substeps is used to increment the change of load or pressure into an equal number of substeps. For example, if a 50 lb point load were added during an event that was controlled by the time KW shown in the example, the load would be applied in 10 lb increments.

Additional Construction Script KW's

CSBC - Indicates that the boundary conditions for a specified list of nodes have changed from the previous time step.

INPUT:

```
KW CSBC
C  node #, X-disp code, Y-disp code, PW code, X-disp, Y-displ, PWP
```

Example:

```
KW CSBC
C  node #, X-disp code, Y-disp code, PW code, X-disp, Y-disp, PWP
   10    1     1   0     0.0     0.250    0.0
   22    1     0   1     0.0     0.0     20.0
```

In the above example, node 10 is fixed in the X and Y directions and the pore water pressure is free to change. A positive Y-displacement of 0.25 units is applied. Node 22 is fixed in the X direction, free to move in the Y-direction, and the pore water pressure is fixed at 20.0 unit of pressure. When a degree of freedom code is 0 (free), the actual value of the parameter written on the CSBC line has no effect on the results, but the field must be filled.

CSBL - Identifies that distributed loads are to be applied to a specified boundary (defined by a list of nodes) during a given time step.

INPUT:

```
KW CSBL
C   node list (cnr node, midside node, cnr node) normal pressure to be
applied
C  at each node
```

Example:

```
KW CSBL
C   node list (cnr node, midside node, cnr node) normal pressure to be
applied
C   at each node
12  23  14  -10  -10  -10
8    0  11    5    5    5
```

In the example, a 10-unit pressure (in units of Patm) is applied normal and to the right of the vector formed by the nodes 12, 23, and 14. A 5-unit pressure is applied to the left of the vector formed by nodes 8 and 11 (no mid-side node on the loaded boundary). Care must be taken to check the

order in which the nodes are entered when a mid-side node is present. Reversing the order of the nodes reverses the direction of the applied loading. The actual loads applied to the nodes are listed in the status reports sent to the output file (also, see KW VBON/VBOF).

CSBP - Lists bar elements which are to be installed during a given time step.
CSBR - Lists bar elements which are to be deleted during a given time step.

Both KW's have the same format. They may be used together within the same time step or used alone. Only CSBR will be illustrated.

INPUT:

KW CSBR
C element number list (up to 8 elements per line)

Example:

KW CSBR
C element number list (up to 8 elements per line)
6 9 12

In the example, bar elements 6, 9, and 12 are removed at the beginning of the time step.

CSBU - Lists elements which are to be submerged or emerged during the current time step.

INPUT:

KW CSBU
C element list (up to 8 elements per line)

Example:

KW CSBU
C element list (up to 8 elements per line)
4,6,9, 8

CSEC - Contains data changing the material type of an element. For example, an element can be excavated during one time step as one material type and filled back in during the current time step as a different type of material.

KW CSEC
C element number, material number (one element per line)

Example:

KW CSEC
C element number, material number (one element per line)
5 2

CSEX - Lists the solid elements which are to be excavated during the current time step.

INPUT:

KW CSEX
C element number list

Example:

KW CSEX
C element number list
7 18 9

CSFP - Indicates that a list of solid elements are to be added during the current time step. This keyword is used for simulation of fill placement.

INPUT:

KW CSFP
C element number list (up to 8 elements per line)

Example:

KW CSFP
C element number list (up to 8 elements per line)
7 8 9

CSFL - Lists beam elements which are to be installed during a given time step.
CSFR - Lists beam elements which are to be deleted during a given time step.

Both KW's have the same format. They may be used together within the same time step or used alone. Only CSFR will be illustrated.

INPUT:

KW CSFR
C element number list (up to 8 elements per line)

Example:

KW CSFR

C element number list (up to 8 elements per line)
6 9 12

In the example, beam elements 6, 9, and 12 are removed at the beginning of the time step.

CSNG - Changes the acceleration due to gravity. This feature is be used to model centrifuge testing.

INPUT:

KW CSNG
C new gravity in units of gravity (g)

Example:

KW CSNG
C new gravity in units of gravity (g)
20.0

In the example the gravity is changed to 20-g. This feature is used to simulate centrifuge testing. The default value is 1.0 g.

CSPL - Identifies concentrated loads are to be applied to a specified nodes during a given time step.

INPUT:

KW CSPL
C node number, point load X-component, point load Y-component

Example:

KW CSPL
C node number, point load X-component, point load Y-component
29 50, -10

In the example, 50 units of force is applied at node 29 parallel to the positive direction (right) on the X-axis and 10 units of force is applied downward at the location of node 29.

CSSP - Lists slip-separation elements which are to be installed during a given time step.
CSSR - Lists slip-separation elements which are to be deleted during a given time step.

Both KW's have the same format. They may be used together within the same time step or used alone. Only CSSR will be illustrated.

INPUT:

KW CSSR
C element number list (up to 8 elements per line)

Example:

KW CSSR
C element number list (up to 8 elements per line)
6 9 12

In the example, slip-separation elements 6, 9, and 12 are removed at the beginning of the time step.

Appendix B
File Structure

The user must provide two files to run STUBBS, a data file and a filename file. The data file contains all the problem information regarding geometry, material parameters, boundary conditions, and initial conditions. The filename file must be named "FEA_FILE." The filename file contains the name of six files used or created by STUBBS. The files may be named using the usual file name conventions found in DOS. That is, the filename may be up to eight letters in length and must not contain certain "reserved" characters. If the program is run on a UNIX system, the file name may be any name acceptable to the UNIX operating system. The order of each file in the list determines the use or content of the file. The "files" list in FEA_FILE is shown in Table B1.

Table B1
Contents of FEA_FILE

FEA_FILE	File Names	File Content
Line 1	TEST.DAT	Input file - provided by user
Line 2	TEST.ERR	Error message file - created by STUBBS
Line 3	TEST.OUT	Output file containing solution - created by STUBBS
Line 4	TEST.MES	Mesh File - created by STUBBS
Line 5	TEST.PLT	Plot File for post processor- created by STUBBS
Line 6	TEST.BAR	Bar Forces - created by STUBBS

It is not required that the files each have the same base name or that extensions be used. The system shown in the example has been found to be very useful by the authors. This naming system has been used with the examples in Appendix C. The STUBBS program uses the first filename (line 1) as the name of the data file. If a user is making multiple runs and wishes to avoid filling disk space with files that may not be needed to evaluate the problem, the user may wish to keep the names of one or more of the program-generated files, the same from one run to the next. The savings in disk space is gained by over-writing the files on each run. If the results of each run is to be saved or used in the postprocessor XMESH, the user should make the file names unique for each run. Be advised that the output file

(named on line 3 of the FEA_FILE) for a large problem (more than several nodes) can be quite lengthy if the verbose KW is on (VBON) and many time steps are used. The contents of the output file are described later in this Appendix.

The user provides a suitable data file (for example, TEST.DAT) and the FEA_FILE. The program will write the remaining five files listed in the FEA_FILE during program execution. The program also writes seven other files for internal use named:

TAPE1
TAPE2
TAPE34
SCRIPT
CSPLR
CSBLR
CSBCR

The file TAPE1 contains a listing of the data file without keywords. TAPE2 contains the data file information with keywords listed followed by the number of data cards associated with the keyword on the same line. TAPE2 will not contain any of the data following the KW LFTC. The file TAPE34 contains the construction script from the data file. SCRIPT contains the construction script as modified by the program to create "new" time steps to accommodate substeps placed in the construction script supplied by the user in the data file. The files CPLR, CSBLR, and CSBCR hold the point load, distributed load, and boundary condition data used in the creation of "new" time steps. See APPENDIX C for an examples of listings of a typical data file and the associated files written by STUBBS.

The Input File

The input file is composed of keywords followed by the data pertaining to the keywords. The keywords are indicated by the letters "KW" followed by one of the four letter keywords. For example, the keyword unit is shown within the input file as:

KW UNIT

In the listing presented in this section, the "KW" indicator is omitted to avoid repetition. The data following the keywords must be separated by one or more spaces, **NOT** a comma.

The input file must contain the following KW's:

UNIT - Specifies units, convergence criteria, and time integration factor
SPEC - Specifies type of problem
NODE - Defines node by number, location, displacement constraints and pore water constraints.

The nodes are formed into elements defined by one or more of the following KW's:

ELEM - Configures solid elements
BEND - Configures bending elements
BREL - Configures bar elements
SLIP - Configures slip-separation elements

The element numbers for those elements present at the start of the problem are specified by one or more of the following KW's:

XBND - Identifies preexisting beam elements
XBRE - Identifies preexisting bar elements
XELT - Identifies preexisting solid elements
XSLP - Identifies preexisting slip elements

The KW's defining existing elements must NOT appear before the KW that form the elements. For example, the XBRE must not appear before BREL or the program will fail to configure the bar elements.

The elements are composed of a material type defined by one or more of the following KW's:

SMPR - Linear elastic particulate properties
ENDO - Endochronic particulate properties
BMMP - Linear elastic (flexural) solid properties
BRMP - Linear elastic bar properties
SLMP - Interface properties

The input file may contain additional information setting the initial conditions at certain nodes or within certain elements using one or more of the following KW's:

IBND - Initial state of stress of preexisting bending elements
IDPP - Initial node data: the prescribed values and boundary conditions of preexisting nodes
ISPR - Provides initial stresses and state data for preexisting solid elements
ISRE - State of preexisting bar elements in terms of stresses
ISSL - Specified displacements for preexisting slip elements

The input file may contain a construction script that is preceded by the KW LFTC and terminated by the KW DONE. Additional construction KW's that may be used are shown below:

CSBC - Indicates that the boundary conditions for a specified list of nodes have changed from the previous time-step. For example, node 100 can be changed from having FIXED-FIXED displacement boundary conditions to FREE-FREE boundary conditions using 'CSBC'.

CSBL - Identifies that distributed loads are to be applied to a specified boundary (defined by a list of nodes) during a given time-step.

CSBP - Lists bar elements which are to be installed during a given time-step.

CSBR - Lists bar elements which are to be deleted during a given time-step.

CSBU - Lists elements which are to be submerged or emerged during the current time-step.

CSEC - Contains data changing the material type of an element. For example, an element can be excavated during one time-step as one material type and filled back in during the current time-step as a different type of material.

CSEX - Lists the solid elements which are to be excavated during the current time-step.

CSFL - Indicates that a list of beam elements are to be added during the current time-step.

CSFP - Indicates that a list of solid elements are to be added during the current time-step.

CSFR - Indicates that a list of beam elements are to be removed during the current time-step.

CSNG - Changes the acceleration due to gravity. This feature is be used to model centrifuge testing.

CSPL - Identifies nodes that receive concentrated loads during a given time-step.

CSSP -Indicates that a list of slip-separation elements are to be added during the current time-step.

CSSR - Indicates that a list of slip-separation elements are to be removed during the current time-step.

TIME - Indicates a construction time-step of a given duration and divided into a number of substeps.

DONE - Indicates the end of a time-step. DONE must terminate the construction script. There must be one DONE line for each KW TIME (time step) and one DONE line to terminate the script closing out the activity initiated by the KW LFTC.

Summary of Keyword Structure

In general, the KW's can be placed in any order without causing errors in execution. It is recommended that the following order be used.

Required KW's:

UNIT
SPEC
NODE

The required keywords must be followed by one or more of the following KW's as shown in Table B2.

Table B2
Keywords Associated with the Various Element Types

Solid Elements	Beam Elements	Bar Elements	Slip-separation Elements	Purpose of KW
ELEM	BEND	BREL	SLIP	Configures elements
XELT	XBND	XBRE	XSLP	Identifies preexisting elements
SMPR ENDO	BMMP	BRMP	SLMP	Provides material properties
IDPP	IDPP	IDPP	IDPP	Provides initial node data
ISPR	IBND	ISRE	ISSL	Provides initial element data

Followed by construction script KW's.

LFTC / DONE
TIME / DONE

Followed by some combination of construction script KW's Cxxx.

For example:

CSPL or CSBC or CSBL could be used in combination, or alone, within a time-step.

Then, each time-step must be terminated with the KW DONE.

And, finally, the script is terminated by KW DONE.

The examples found in Appendix C illustrate the proper input file structure for an assortment of typical problems.

The Output Files

STUBBS produces 11 files as output. Some of these are scratch files that have little interest for the user. A listing of the files is provided in Table B3. The output file (*.out), the plot file (*.plt), and the error file (*.err) provide vital information to the user. The contents of these three files is described in the following sections.

Table B3
Files Written by STUBBS

File Name.EXT	Contents
anyname.ERR	Error messages generated within STUBBS
anyname.OUT	Results of calculations and status reports
anyname.MES	Mesh data written by STUBBS
anyname.PLT	Data for postprocessing with XMESH
anyname.BAR	Data for bar elements
SCRIPT	Modified script file
CSPLR	Point load file
CSBLR	Distributed load file
CSBCR	Boundary conditions file
TAPE1	INPUT scratch file (KW stripped)
TAPE2	INPUT scratch file No. 2 (KW shown)
TAPE34	Construction script file

The *.OUT file

The file with an OUT extension is the primary source of information to the user. The file contains three types of information: Initial Status Report, Construction Activity Status Data, and Results of Calculations. The construction activity status reports are turned off and on by KW VBOF and VBON, respectively. The default is VBON. The results of calculations reports are turned off and on by KW RPOF and RPON, respectively. The default is RPON. The initial status report is always placed in the *.OUT file and can not be suppressed.

Initial status report

The initial status report contains ten sections for either deformation or coupled flow problems, and five sections for pure flow problems. The contents of the sections are listed in Table B4. The Initial Status Report is always generated. Flow problems produce less initial data because the

primary output of a pure flow problem are flow velocities. Flow velocities are computed from pressures after the first construction time-step. STUBBS does not initialize flow velocities and does not permit initial velocities to be entered in the input file.

Table B4
Initial Status Report

Section Name	Section Contents
Problem summary	Title, time/date of run, itemization of nodes and elements, number of time steps, physical unit values selected in KW UNIT, maximum band width, number of equations, coefficient matrix size, and convergence criteria.
Nodal point data	Node numbers, coordinates, DOF's, initial displacement and PWP, and an extra column of data (that is not currently used, zeros shown in file).
Connectivity data	Tables showing element number, material number, and nodes. Tables for all types of elements used are provided.
Initial values in elements	Table listing element numbers, σ_{XX} , σ_{YY} , τ_{XY} , void ratio, and degree of saturation.
Material property data	An echo of the material property data for all type of elements used. In flow and coupled flow problems, a table of pressure-saturation data for each solid material follows the echo of material properties.
Summary of active elements	Table of all preexisting elements listing their status as "new" and the relationship as above or below water table for solid elements. Global element numbers are used instead of element type number. **Omitted in pure flow problems.**
Summary of active nodes	Table listing node number, status = new, DOF's, FX, FY. **Omitted in pure flow problems.**
Residual load data / saturation data	Line stating "Residual Load Factor =" xx.xx "after" n "iterations," followed by a table of element numbers with the degree of saturation computed for each of the four gauss points (averaging points) within the element. **Omitted in pure flow problems.**
Displacements and pore pressures for T = 0.0	Table listing node number, displacements, and pwp. **Omitted in pure flow problems.**
Summary of stresses at element centers	Table listing element number, σ_X ,σ_Y ,σ_Z , τ_{XY} , pwp, F. (F is not currently used, zeros show in file. **Omitted in pure flow problems.**

Construction activity status data

During a time-step, pore water pressure DOF's are automatically turned on or off as a result of a rising or falling water table, boundary loadings, excavation or filling. The user can track the status of nodes and elements using the construction activity status data. Nodes are assigned a status depending on their situation at the start of a time-step. The possible status labels are surface boundary (SB), permanent boundary (PB), excavated

boundary (EB), and interior (IN). The data is composed of three sections shown in Table B5.

Table B5 Construction Activity Status Data	
Section Name	**Section Contents**
Summary of activities	A list of the following data Total time-step = Number of increments = Total time at end of step = Number of equations = Stiffness size (bytes) = An announcement of the number of boundary conditions changed by KW CSBC, the number of elements added or removed, and a the number of loadings added or removed.
Summary of active elements	Table of all existing elements (for the particular time step) listing their status as new or old and the relationship as above or below water table for solid elements. The globals element numbers are used in the summary table. Element type numbers are used in all other tables.
Summary of active nodes	Table listing node number, status = new/ old , status label, DOF's, FX, FY.

Results of calculations

The calculated results are written to the output file for each time step unless the report is suppressed by KW RPOF. The user can control the selective reporting of results for specific time-steps by using RPOF and RPON in combinations. The information is written in sections shown in Table B6.

Table B6 Results of Calculations	
Section Name	**Section Contents**
Residual load data	Line stating "Residual Load Factor =" xx.xx "after" n "iterations"
Displacements and pore pressures for T = xxx.xx	Table listing node number, displacements, and pwp. **Omitted in pure flow problems.**
Flow velocities and pressures for T = xxx.xx	Table listing node number, velocities in x and y directions, pwp, and total head. **Omitted in pure displacement problems.**
Summary of stresses at element centers	Table listing element number, σ_X ,σ_Y ,σ_Z , τ_{XY} , pwp, F. The "F" parameter is used only with plastic soil materials. For linear elastic problems, zeros fill the "F" column.

The *.PLT file

The plot file contains data that can be brought into XMESH, a post-processor developed specifically for STUBBS. The plot data is generally the information directed to the output file as the results of calculation data, but without titles.

The first line of the plot file contains four numbers: the number of nodes, the number of elements, the number of additional node parameters, and the number of additional element parameters. The additional parameters are in excess of the basic node and element parameters. The basic parameters depend on the problem type. For deformation and coupled flow, the basic node parameters are node number, x-coordinate, and y-coordinate. For flow problems, the basic node parameters are node number, flow velocity in x-direction, and flow velocity in y-direction.

An additional node parameter for deformation and coupled flow is pore water pressure. For pure flow, three additional node parameters are placed into the plot file, total head, pore water pressure, and a dry-wet code.

The basic element parameters are σ_X ,σ_Y, and τ_{XY}. Additional element parameters for deformation and coupled flow problems are pore water pressure and a parameter "F" (mobilized shear strength factor, used only for plastic soil (KW ENDO)). For pure flow problems, the additional element parameters are pore water pressure, saturation (shown as a decimal), and water content.

The names of these additional parameters follow the first line with the names of the node parameters preceding the names of the element parameters. The basic parameters are not named within the plot file. The first lines of typical plot files are shown in Table B7.

Table B7
First Lines of Typical Plot Files

Pure Deformation	Deformation and Coupled Flow	Pure Flow
35 24 1 2 PWP PWP F	35 24 5 2 T-head VEL_X VEL_Y Dry-Wet PWP S WC	35 24 3 3 T_head PWP Dry-Wet PWP S WC

Node and element data follow the labels. The node and element data must be preceded by a line that provides the time associated with the end of the time step. Usually, the initial time is zero except in pure flow problems. The time data line appears as:

T = 0.0

Then, the node and element data follow. Each node and element is listed consecutively on a line that contains the basic and additional data in columns separated by spaces. Data for additional times follow at the end of the element data. Finally, the file is terminated by an END line.

The plot file can serve as a quick source of information to an experienced user. The plot file is often much smaller than the output file due to the absence of headings.

The *.ERR file

The error file contains error messages that are generated by the error trapping done by STUBBS. The error file can be helpful in correcting mistakes, but serious errors generally cause STUBBS to abort. When serious problems occur, error messages are sent to the screen by the compiler or operating system, and none of the 10 files written by STUBBS are generated.

Appendix C
Examples

This appendix is a comprehensive set of examples that demonstrate the proper file format and output for a wide variety of problems. The examples are cross referenced by problem type, element type, and keyword. In addition, an alphabetical listing of problem filenames is provided.

In general, these problems have plane strain conditions. Five axisymmetric examples are provided in files fourax.dat, ate_ax1.dat, cateax1.dat, cateaxr.dat, and fourendo.dat. Three coupled flow problems are provided in files cate_1.dat, cateax1.dat, and cateaxr.dat. Two pure flow problems are provided in files "flow1.dat" and "flow2.dat."

The first example in the appendix has a complete listing of both input and output files. It is recommended that a new user review the first example (fours1.dat) in detail before attempting any other example provided in this appendix.

Examples by element type:

Table C1
Plane Strain - Pure Deformation Examples

Solid (Basic Mesh Series)	Ending	Bar	Slip-separation
fours1.dat sixs1.dat sixs2.dat ate_1.dat trans1.dat	beam_t10.dat	truss1.dat roark1.dat	reed2a.dat reed2b.dat

Note that all of the solid element examples used the same collection of nodes and are collectively called the **basic mesh series**. The configuration of the elements was different in each problem. Fours1 used twenty four 4-node elements, Sixs1 and sixs2 used twelve 6-node solid elements. In sixs1, the elements were oriented with the 3-node sides vertical; in sixs2, the 6-node elements were oriented with the 3-node sides horizontal. File ate_1 used six

8-node elements. File trans1 used two 8-node, two 5-node, and two 4-node elements.

The alphabetical list of examples is provided in Table C2. The index of keywords and examples is given in Table 3 following the alphabetical listing.

Table C2
Alphabetical Listing of Examples

Filename	Description	Remarks	Page
ate_1	8-node solid block with distributed boundary loads	Basic Mesh Series (BMS)	C18
ate_ax1	8-node axisymmetric rod with axial loading	BMS	C21 - C22
beam_t10	cantilever beam composed of 10 elements		C23 - C26
cateax1	axisymmetric 8-node solid block consolidating under distributed boundary loads	BMS top drainage only	C41 - C42
cateaxr	axisymmetric 8-node solid block consolidating under distributed boundary loads	BMS top and radial drainage	C41 - C42
cate_1	8-node solid block consolidating under distributed boundary loads	BMS	C41 - C43
flow1	constant head test - two soils in series	BMS	C34 - C40
flow2	constant head test - two soils in parallel	BMS	C39 - C40
fourendo	4-node elements in an axisymmetric column composed of plastic material	BMS	C43 - C45
fours1	4-node solid block with distributed boundary loads	BMS	C6 - C18
fours1ax	4-node axisymmetric column with axial loading	BMS	C23
reed2a	2 solid blocks	Simulates ∞ compliance	C31 - C34
reed2b	slip elements between 2 solid blocks		C31 - C33
roark1	complex truss (bar elements)		C28 - C30
sixs1	6-node elements (horizontal) solid blocks with boundary loads	BMS	C18 - C19
sixs2	6-node elements (vertical) solid blocks with boundary loads	BMS	C19
trans1	5-node solid block transition element with boundary loads	BMS	C20
truss1	simple truss (bar elements)		C26 - C28

Table C3
Cross-Reference Keyword to Example Input Files

Keyword	Examples Featuring the Keyword
UNIT	All - required KW
SPEC	All - required KW
NODE	All - required KW
ELEM	All Basic Mesh Series, reed2a, reed2b
BEND	beam t10
BREL	truss1, roark1
SLIP	reed2b
TIME / DONE	All
LFTC / DONE	All
XELT	All Basic Mesh Series, reed2a, reed2b
XBND	beam t10
XBRE	truss1, roark1
XSLP	reed2b
SMPR	All Basic Mesh Series, reed2a, reed2b
ENDO	fourendo
BMMP	beam t10
BRMP	truss1, roark1
SLMP	reed2b
IDPP	cate 1, cateax, cateaxr, flow1, flow2
CSBC	flow1, flow2, cate 1, cateax1, cateaxr
CSBL	All Basic Mesh Series, reed2b
CSEC	flow2
CSPL	beam t10, truss1, roark1

The keywords in the table are all of the keywords used in the examples, but are not all possible keywords. The keywords in the examples are among those most frequently used. A complete listing and syntax for all keywords is provided in Appendix A.

Basic Mesh Series

A simple problem is used to illustrate the file formats and typical output. The problem is analyzed using a variety of solid element configurations. The problem is that of a block of weightless material subjected to a uniform load

normal to the top boundary. The displacement of the top boundary is determined and compared to a closed form solution.

Closed form solution - general

The STUBBS code solves two-dimensional problems. The two dimensions can be plane strain or axisymmetric problems. The Basic Mesh Series topic is a plane strain problem. The coordinate system expressed in terms of x ,y, and z, has the y-axis oriented vertically and the z-axis is perpendicular to the page. The problem geometry of plane strain problems is such that one dimension of the problem is very long as compared to the other two dimensions. In our case, the long dimension is parallel to the z-axis. The problem geometry is described completely within the x-y plane. Each x-y slice is identical to all of the other x-y slices, so that any displacement in the z-direction is prohibited by the adjacent slices. That is, no strain is permitted in the z-direction. Stress-strain relations for this situation are:

$$\sigma_z = \mu(\sigma_x + \sigma_y)$$

$$E\,\epsilon_x = [(1-\mu^2)\sigma_x - \mu(1+\mu)\sigma_y]$$

$$E\,\epsilon_y = [(1-\mu^2)\sigma_y - \mu(1+\mu)\sigma_x]$$

$$E\,\gamma_{xy} = 2(1+\mu)\,\tau_{xy}$$

Closed form solution - basic mesh series

In the example problems using solid elements that are part of the collection entitled BASIC MESH SERIES, a block of linear-elastic, weightless, material (4-in. wide and 12-in. high) is subjected to a uniform vertical stress of 10 psi along its top boundary. The expected vertical displacement of the top boundary is uniformly down in the direction of the applied loading. In this situation, σ_x is zero at the boundary of the block. At the base, the block is free to move laterally, but not vertically. Actually, one of the nodes at the bottom of the block was also fixed in the x-direction to ensure stability.

Given:

> $E = 1000$ psi $\mu = 0.3333$ boundary pressure $= 10$ psi and block height (L) $= 12$ in.

Then:

> the displacement (vertical deformation of the top of the block) is found as

$$\Delta = (1-\mu^2)\sigma_y\,L\,/\,E \quad \text{or} \quad 0.8889\,(10)\,(12)\,/\,1000 = 0.1067 \text{ in} \downarrow$$

Using the sign convention of the program, both the applied loading and the resulting vertical displacement have a negative sign in the data file and output file.

Mesh configurations

The problem was described by dividing the block into small elements. STUBBS allows a solid element to have from as few as three nodes, to as many as eight nodes. This problem was solved several times using a variety of mesh configurations based on the same set of nodes. The nodes are shown in Figure C1. The 35 nodes are arranged in seven rows of 5 nodes. These nodes were used to form 24 four-node elements, 12 six-node elements, and 6 eight-node elements. In addition, one used a group of 2 four-node element supporting 2 five-node elements that, in turn, supported 2 eight-node elements. The element configurations are shown in Figure C2.

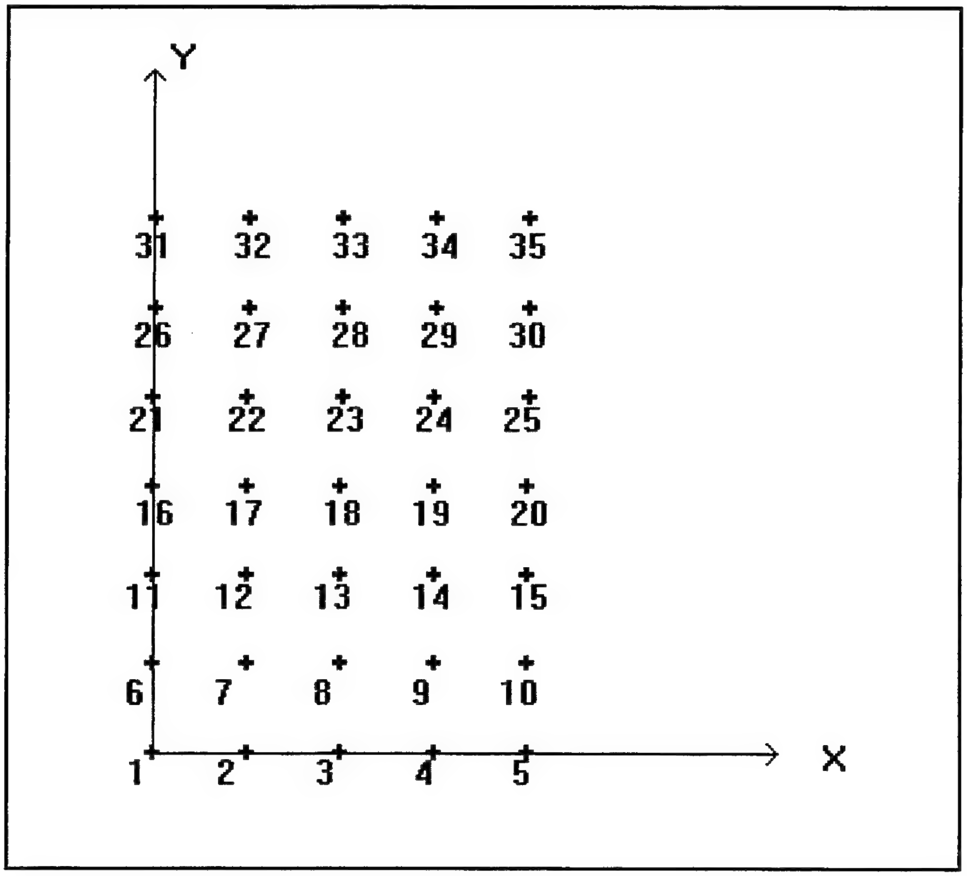


Figure C1. Basic mesh series - nodes

The six-node elements were used in two problems. The orientation of the elements was vertical in one case, and horizontal in the other. The six-node

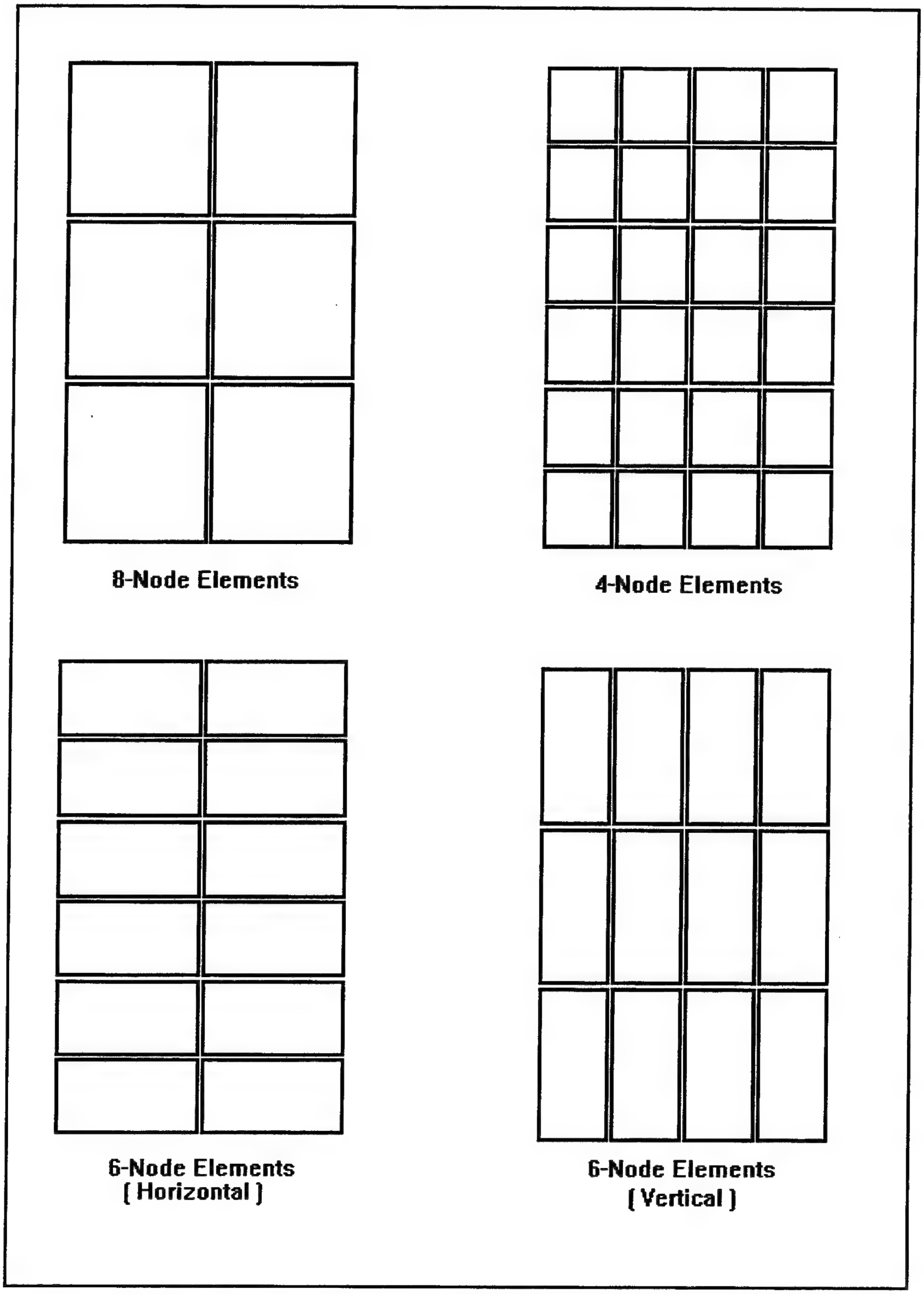


Figure C2. Basic mesh series - elements

elements are composed of 2 three-node sides and 2 two-node sides, with the four corner nodes common to a three-node side and a two-node side. The six-node elements were arranged in one grouping with the three-node sides oriented vertically, and another grouping with the three-node sides oriented horizontally.

The eight data files are named as follows::

fours1.dat	24 elements
flow1.dat	24 elements (discussed later, under flow problems)
flow2.dat	24 elements (discussed later, under flow problems)
sixs1.dat	12 elements (three-node side horizontal)
sixs2.dat	12 elements (three-node side vertical)
ate_1.dat	6 elements
cate_1.dat	6 elements (discussed later, under coupled flow-deformation heading)
trans1.dat	6 elements (2 eight-node, 2 five-node, and 2 four-node)

The five nodes that make up the top of the block are numbered 31 through 35. The displacement of these nodes can be found in the output (file-name.OUT) file. The load was applied using a construction script.

The loading was specified using the KW CSBL. The format of CSBL varies with the nature of the boundary. That is, a distributed loading applied to a boundary composed of two-node sides is coded differently from the case when the boundary is composed of three-node sides. More discussion on the format of CSBL is presented in a later section.

The program provides the following answers:

Table C4
Vertical Displacement Along the Top Boundary

Node numbers	31	32	33	34	35
4-node solid fours1.dat	0.1067	0.1067	0.1067	0.1067	0.1067
6-node solid (horizontal) sixs1.dat	0.1067	0.1067	0.1067	0.1067	0.1067
6-node solid (vertical) sixs2.dat	0.1067	0.1067	0.1067	0.1067	0.1067
8-node solid ate_1.dat, cate_1.dat **	0.1067	0.1067	0.1067	0.1067	0.1067
5-node transition trans1.dat	0.1067	0.1067	0.1067	0.1067	0.1067
Hand solution	0.1067	0.1067	0.1067	0.1067	0.1067

Note that using the program sign convention, all of the answers were negative, indicating downward movement in the direction of the applied boundary loading. The negative signs were omitted for clarity in preparing the table. The displacements have units of inches.

** The coupled flow problem were accurate to three places to the right of the decimal. For further discussion of cate_1, see the section on coupled flow - deformation.

Four-Node Elements

The input file for, fours1.dat, the four-node element mesh follows:

```
CommeSTUBBs Hand Mesh # 2  4-node elements
C  July 25, 1995
C  filename = FOURS1.DAT
C  example using zero unit weight and 10 psi boundary load along top
C  of specimen (plain strain)
KW UNIT
C  ATMOSPHERIC PRESSURE, GAMMA H2O, VALUE OF 1-g (3 required
parameters)
C     psi,                    pci,        units of g
      14.7                    0.0361         1.0
C
KW SPEC
C FLOW, DFORM, DRAIN, CHECK_MESH, CHECK_SCRIPT, AXIS (6)
    F      T       T         F            F           F
C
KW XELT
C  Elements existing at the start of the problem
   1:24
KW SMPR
C  linear elastic material properties
C MAT#,   GAMMAT, RVOID, GS, RSAT (5)
   1      0.00    0.55    2.7   .80
C MODULUS,   POISX,   POISY,    XN, XM,   KO (6)
  1000     .3333  .3333     1.0  0.4  0.5
C PERMX, PERMY, B-Value  (3)
  .001    .001    .999
C van Genuchten parameters (2)
  0.006   2.3
C  End of material properties
C
KW NODES
C   Scaling parameters X-factor, Y-factor
    1.0      1.0
C  Node    X-coord      Y-coord    X      Y     PWP dof's
    1      0.00         0.00       1      1      0
    2      1.0          0.000      0      1      0
    3      2.0          0.000      0      1      0
    4      3.00         0.000      0      1      0
    5      4.00         0.000      0      1      0
    6      0.00         2.000      0      0      0
    7      1.000        2.000      0      0      0
    8      2.000        2.000      0      0      0
    9      3.000        2.000      0      0      0
   10      4.000        2.000      0      0      0
   11      0.000        4.000      0      0      0
   12      1.000        4.000      0      0      0
   13      2.000        4.000      0      0      0
   14      3.000        4.000      0      0      0
   15      4.000        4.000      0      0      0
   16      0.000        6.000      0      0      0
   17      1.000        6.000      0      0      0
   18      2.000        6.000      0      0      0
   19      3.000        6.000      0      0      0
   20      4.000        6.000      0      0      0
```

```
     21      0.000        8.000      0      0      0
     22      1.000        8.000      0      0      0
     23      2.000        8.000      0      0      0
     24      3.000        8.000      0      0      0
     25      4.000        8.000      0      0      0
     26      0.000       10.000      0      0      0
     27      1.000       10.000      0      0      0
     28      2.000       10.000      0      0      0
     29      3.000       10.000      0      0      0
     30      4.000       10.000      0      0      0
     31      0.000       12.000      0      0      0
     32      1.000       12.000      0      0      0
     33      2.000       12.000      0      0      0
     34      3.000       12.000      0      0      0
     35      4.000       12.000      0      0      0
C
KW ELEM
C    4-node connectivity table
C Ele#  Mat #      4 corner nodes              mid-side nodes
     1     1     1     2     7     6     0     0     0     0
     2     1     2     3     8     7     0     0     0     0
     3     1     3     4     9     8     0     0     0     0
     4     1     4     5    10     9     0     0     0     0
     5     1     6     7    12    11     0     0     0     0
     6     1     7     8    13    12     0     0     0     0
     7     1     8     9    14    13     0     0     0     0
     8     1     9    10    15    14     0     0     0     0
     9     1    11    12    17    16     0     0     0     0
    10     1    12    13    18    17     0     0     0     0
    11     1    13    14    19    18     0     0     0     0
    12     1    14    15    20    19     0     0     0     0
    13     1    16    17    22    21     0     0     0     0
    14     1    17    18    23    22     0     0     0     0
    15     1    18    19    24    23     0     0     0     0
    16     1    19    20    25    24     0     0     0     0
    17     1    21    22    27    26     0     0     0     0
    18     1    22    23    28    27     0     0     0     0
    19     1    23    24    29    28     0     0     0     0
    20     1    24    25    30    29     0     0     0     0
    21     1    26    27    32    31     0     0     0     0
    22     1    27    28    33    32     0     0     0     0
    23     1    28    29    34    33     0     0     0     0
    24     1    29    30    35    34     0     0     0     0
C
KW LFTC
C  Start of construction script
KW TIME
C # of substeps, duration
   1   1.
KW CSBL
C Distributed load
C  Nodes         Pressure
  31  0  32  -10 -10 -10
  32  0  33  -10 -10 -10
  33  0  34  -10 -10 -10
  34  0  35  -10 -10 -10
KW DONE
KW DONE
```

Comments concerning the file:

a. Note that the units were set in KW UNIT as pounds and inches.

b. This is a pure deformation problem as the logical variables FLOW and DFORM were set to FALSE and TRUE, respectively. The problem was specified as drained deformation in KW SPEC. The other logical variables were set to false.

c. The modulus was 1,000 psi and Poisson's ratio was 0.333. Permeability parameters were *required* to be provided although the problem did not concern flow.

d. The bottom nodes, number 1 through 5, were fixed in the Y-direction and node 1 was fixed in the X-direction using the KW NODE.

e. The elements were all composed on material 1 as set using KW ELEM.

f. All 24 elements were in-place at the beginning of the problem as set by KW XELT.

g. A construction script was begun by KW LFTC and terminated by KW DONE (last line in input file).

h. A time-step having only one substep and a duration of 1 sec was established by KW TIME. Note that the unit of time was established by the units used for permeability in KW SMPR. However, this is a time independent problem as FLOW is set to FALSE under KW SPEC.

i. The boundary loading of 10 psi was set by the KW CSBL. Note that the boundary is composed of two-node sides of the four-node element and that the midside node in CSBL was set to zero. Contrast the format to that used for CSBL when using the eight-node element in ate_1.dat.

j. Note that KW DONE on the next to last line serves to end the time-step established by KW TIME.

The output file for the four-node mesh follows:

```
                    *** PROBLEM SUMMARY ***

Title:  STUBBs Hand Mesh # 2 4-node elements
Reference date and time: 07/25/95       10:38
Total number of nodes ......................   35
Total number of soil elements ..............   24
Number of diff. soil materials .............    1
Number of bar elements .....................    0
Number of different bar materials ..........    0
Number of interface elements ...............    0
Number of interface types  .................    0
Number of bending elements .................    0
Number of beam types  ......................    0
Number of preexisting bar elements .........    0
Number of preexisting interface elements ...    0
Number of preexisting bending elements .....    0
Number of preexisting elements .............   24
Total number of elements ...................   24
Number of load cases .......................    1
Convergence criterion ...................... 0.1000E-09
   Within specified number of iterations ...   10
Time integration factor, Theta .............. .667

PHYSICAL UNITS:
   Atmospheric pressure .....    14.7000
   Acceleration of gravity ..     1.0000
   Unit weight of water .....     0.0361
```

FOR COMPLETE SYSTEM:

Maximum band width	13
Number of equations	64
Coefficient matrix	751

*** NODAL POINT DATA ***

Node	Nodal Coordinates		Boundary Code			Initial Values			
Number	X (horz)	Y (vert)	X	Y	U	Disp X	Disp Y	PWP	
1	0.000	0.000	fx	fx	fx	0.0000	0.0000	0.0000	0.0000
2	1.000	0.000		fx	fx	0.0000	0.0000	0.0000	0.0000
3	2.000	0.000		fx	fx	0.0000	0.0000	0.0000	0.0000
4	3.000	0.000		fx	fx	0.0000	0.0000	0.0000	0.0000
5	4.000	0.000		fx	fx	0.0000	0.0000	0.0000	0.0000
6	0.000	2.000			fx	0.0000	0.0000	0.0000	2.000
7	1.000	2.000			fx	0.0000	0.0000	0.0000	2.000
8	2.000	2.000			fx	0.0000	0.0000	0.0000	2.000
9	3.000	2.000			fx	0.0000	0.0000	0.0000	2.000
10	4.000	2.000			fx	0.0000	0.0000	0.0000	2.000
11	0.000	4.000			fx	0.0000	0.0000	0.0000	4.000
12	1.000	4.000			fx	0.0000	0.0000	0.0000	4.000
13	2.000	4.000			fx	0.0000	0.0000	0.0000	4.000
14	3.000	4.000			fx	0.0000	0.0000	0.0000	4.000
15	4.000	4.000			fx	0.0000	0.0000	0.0000	4.000
16	0.000	6.000			fx	0.0000	0.0000	0.0000	6.000
17	1.000	6.000			fx	0.0000	0.0000	0.0000	6.000
18	2.000	6.000			fx	0.0000	0.0000	0.0000	6.000
19	3.000	6.000			fx	0.0000	0.0000	0.0000	6.000
20	4.000	6.000			fx	0.0000	0.0000	0.0000	6.000
21	0.000	8.000			fx	0.0000	0.0000	0.0000	8.000
22	1.000	8.000			fx	0.0000	0.0000	0.0000	8.000
23	2.000	8.000			fx	0.0000	0.0000	0.0000	8.000
24	3.000	8.000			fx	0.0000	0.0000	0.0000	8.000
25	4.000	8.000			fx	0.0000	0.0000	0.0000	8.000
26	0.000	10.000			fx	0.0000	0.0000	0.0000	10.00
27	1.000	10.000			fx	0.0000	0.0000	0.0000	10.00
28	2.000	10.000			fx	0.0000	0.0000	0.0000	10.00
29	3.000	10.000			fx	0.0000	0.0000	0.0000	10.00
30	4.000	10.000			fx	0.0000	0.0000	0.0000	10.00
31	0.000	12.000			fx	0.0000	0.0000	0.0000	12.00
32	1.000	12.000			fx	0.0000	0.0000	0.0000	12.00
33	2.000	12.000			fx	0.0000	0.0000	0.0000	12.00
34	3.000	12.000			fx	0.0000	0.0000	0.0000	12.00
35	4.000	12.000			fx	0.0000	0.0000	0.0000	12.00

* => Preexisting

*** SOIL ELEMENT CONNECTIVITY DATA ***

Element	Material	Nodes							
Number	Type	I	II	III	IV	V	VI	VII	VIII
1*	1	1	2	7	6	0	0	0	0
2*	1	2	3	8	7	0	0	0	0
3*	1	3	4	9	8	0	0	0	0
4*	1	4	5	10	9	0	0	0	0
5*	1	6	7	12	11	0	0	0	0
6*	1	7	8	13	12	0	0	0	0
7*	1	8	9	14	13	0	0	0	0
8*	1	9	10	15	14	0	0	0	0
9*	1	11	12	17	16	0	0	0	0
10*	1	12	13	18	17	0	0	0	0
11*	1	13	14	19	18	0	0	0	0
12*	1	14	15	20	19	0	0	0	0
13*	1	16	17	22	21	0	0	0	0
14*	1	17	18	23	22	0	0	0	0
15*	1	18	19	24	23	0	0	0	0

16*	1	19	20	25	24	0	0	0	0
17*	1	21	22	27	26	0	0	0	0
18*	1	22	23	28	27	0	0	0	0
19*	1	23	24	29	28	0	0	0	0
20*	1	24	25	30	29	0	0	0	0
21*	1	26	27	32	31	0	0	0	0
22*	1	27	28	33	32	0	0	0	0
23*	1	28	29	34	33	0	0	0	0
24*	1	29	30	35	34	0	0	0	0

* => Preexisting

*** INITIAL VALUES IN SOLID ELEMENTS ***

Element	Sig X	Sig Y	Tau XY	VOID	Degree of Sat
1	0.000	0.000	0.000	0.550	0.800
2	0.000	0.000	0.000	0.550	0.800
3	0.000	0.000	0.000	0.550	0.800
4	0.000	0.000	0.000	0.550	0.800
5	0.000	0.000	0.000	0.550	0.800
6	0.000	0.000	0.000	0.550	0.800
7	0.000	0.000	0.000	0.550	0.800
8	0.000	0.000	0.000	0.550	0.800
9	0.000	0.000	0.000	0.550	0.800
10	0.000	0.000	0.000	0.550	0.800
11	0.000	0.000	0.000	0.550	0.800
12	0.000	0.000	0.000	0.550	0.800
13	0.000	0.000	0.000	0.550	0.800
14	0.000	0.000	0.000	0.550	0.800
15	0.000	0.000	0.000	0.550	0.800
16	0.000	0.000	0.000	0.550	0.800
17	0.000	0.000	0.000	0.550	0.800
18	0.000	0.000	0.000	0.550	0.800
19	0.000	0.000	0.000	0.550	0.800
20	0.000	0.000	0.000	0.550	0.800
21	0.000	0.000	0.000	0.550	0.800
22	0.000	0.000	0.000	0.550	0.800
23	0.000	0.000	0.000	0.550	0.800
24	0.000	0.000	0.000	0.550	0.800

*** MATERIAL PROPERTY DATA ***

Material Number: 1
Material Type: SOIL

```
Weight-Volume Relationships:
  Unit Weight ................................ 0.000
  Reference Void Ratio ....................... 0.550
  Reference Degree of Saturation ............. 0.800
  Specific Gravity ........................... 2.70

Elastic Properties:
   Vertical Youngs Modulus ................... 0.100E+04
   Poisson's Ratio (x-z plane) ............... 0.3333
   Poisson's Ratio (x-y plane) ............... 0.3333
   Horizontal to Vertical Stiffness Ratio .... 1.000
   Shear to Vertical Stiffness Ratio ......... 0.4000
   Coefficient of Earth Pressure at Rest ..... 0.5000

Flow Properties:
  Horizontal Saturated Permeability .......... 0.100E-02
  Vertical Saturated Permeability ............ 0.100E-02
  Skempton's B Parameter ..................... 0.999

Van Genuchten Parameters:
   Alpha ..................................... 0.600E-02
   n ......................................... 2.30
   m ......................................... 0.565
```

SUMMARY OF DATA FOR INITIAL ELEMENTS

*** SUMMARY OF ACTIVE ELEMENTS ***

Elem	Status		
1	Soil No. 1 new fill	below water table	recompute body force
2	Soil No. 1 new fill	below water table	recompute body force
3	Soil No. 1 new fill	below water table	recompute body force
4	Soil No. 1 new fill	below water table	recompute body force
5	Soil No. 1 new fill	below water table	recompute body force
6	Soil No. 1 new fill	below water table	recompute body force
7	Soil No. 1 new fill	below water table	recompute body force
8	Soil No. 1 new fill	below water table	recompute body force
9	Soil No. 1 new fill	below water table	recompute body force
10	Soil No. 1 new fill	below water table	recompute body force
11	Soil No. 1 new fill	below water table	recompute body force
12	Soil No. 1 new fill	below water table	recompute body force
13	Soil No. 1 new fill	below water table	recompute body force
14	Soil No. 1 new fill	below water table	recompute body force
15	Soil No. 1 new fill	below water table	recompute body force
16	Soil No. 1 new fill	below water table	recompute body force
17	Soil No. 1 new fill	below water table	recompute body force
18	Soil No. 1 new fill	below water table	recompute body force
19	Soil No. 1 new fill	below water table	recompute body force
20	Soil No. 1 new fill	below water table	recompute body force
21	Soil No. 1 new fill	below water table	recompute body force
22	Soil No. 1 new fill	below water table	recompute body force
23	Soil No. 1 new fill	below water table	recompute body force
24	Soil No. 1 new fill	below water table	recompute body force

*** SUMMARY OF ACTIVE NODES ***

Node	Status	DOF-X	DOF-Y	DOF-PWP	FX	FY
1 New	Permanent boundary	Fixed	Fixed	NOFLO	0.0000	0.0000
2 New	Permanent boundary	1	Fixed	NOFLO	0.0000	0.0000
3 New	Permanent boundary	2	Fixed	NOFLO	0.0000	0.0000
4 New	Permanent boundary	3	Fixed	NOFLO	0.0000	0.0000
5 New	Permanent boundary	4	Fixed	NOFLO	0.0000	0.0000
6 New	Surface boundary	5	6	NOFLO	0.0000	0.0000
7 New	Interior	7	8	NOFLO	0.0000	0.0000
8 New	Interior	9	10	NOFLO	0.0000	0.0000
9 New	Interior	11	12	NOFLO	0.0000	0.0000
10 New	Surface boundary	13	14	NOFLO	0.0000	0.0000
11 New	Surface boundary	15	16	NOFLO	0.0000	0.0000

12	New	Interior	17	18	NOFLO	0.0000	0.0000
13	New	Interior	19	20	NOFLO	0.0000	0.0000
14	New	Interior	21	22	NOFLO	0.0000	0.0000
15	New	Surface boundary	23	24	NOFLO	0.0000	0.0000
16	New	Surface boundary	25	26	NOFLO	0.0000	0.0000
17	New	Interior	27	28	NOFLO	0.0000	0.0000
18	New	Interior	29	30	NOFLO	0.0000	0.0000
19	New	Interior	31	32	NOFLO	0.0000	0.0000
20	New	Surface boundary	33	34	NOFLO	0.0000	0.0000
21	New	Surface boundary	35	36	NOFLO	0.0000	0.0000
22	New	Interior	37	38	NOFLO	0.0000	0.0000
23	New	Interior	39	40	NOFLO	0.0000	0.0000
24	New	Interior	41	42	NOFLO	0.0000	0.0000
25	New	Surface boundary	43	44	NOFLO	0.0000	0.0000
26	New	Surface boundary	45	46	NOFLO	0.0000	0.0000
27	New	Interior	47	48	NOFLO	0.0000	0.0000
28	New	Interior	49	50	NOFLO	0.0000	0.0000
29	New	Interior	51	52	NOFLO	0.0000	0.0000
30	New	Surface boundary	53	54	NOFLO	0.0000	0.0000
31	New	Surface boundary	55	56	NOFLO	0.0000	0.0000
32	New	Surface boundary	57	58	NOFLO	0.0000	0.0000
33	New	Surface boundary	59	60	NOFLO	0.0000	0.0000
34	New	Surface boundary	61	62	NOFLO	0.0000	0.0000
35	New	Surface boundary	63	64	NOFLO	0.0000	0.0000

* = Side node with no pore pressure dof.
NOFLO >> FLOW is FALSE during initial calculations

RESIDUAL = 0.0000 after 1 iterations.

DISPLACEMENTS AND PORE PRESSURES FOR T = 0.0000

Node	Disp x	Disp y	PWP
1	0.0000	0.0000	0.0000
2	0.0000	0.0000	0.0000
3	0.0000	0.0000	0.0000
4	0.0000	0.0000	0.0000
5	0.0000	0.0000	0.0000
6	0.0000	0.0000	0.0000
7	0.0000	0.0000	0.0000
8	0.0000	0.0000	0.0000
9	0.0000	0.0000	0.0000
10	0.0000	0.0000	0.0000
11	0.0000	0.0000	0.0000
12	0.0000	0.0000	0.0000
13	0.0000	0.0000	0.0000
14	0.0000	0.0000	0.0000
15	0.0000	0.0000	0.0000
16	0.0000	0.0000	0.0000
17	0.0000	0.0000	0.0000
18	0.0000	0.0000	0.0000
19	0.0000	0.0000	0.0000
20	0.0000	0.0000	0.0000
21	0.0000	0.0000	0.0000
22	0.0000	0.0000	0.0000
23	0.0000	0.0000	0.0000
24	0.0000	0.0000	0.0000
25	0.0000	0.0000	0.0000
26	0.0000	0.0000	0.0000
27	0.0000	0.0000	0.0000
28	0.0000	0.0000	0.0000
29	0.0000	0.0000	0.0000
30	0.0000	0.0000	0.0000
31	0.0000	0.0000	0.0000
32	0.0000	0.0000	0.0000
33	0.0000	0.0000	0.0000
34	0.0000	0.0000	0.0000
35	0.0000	0.0000	0.0000

SUMMARY OF STRESSES AT ELEMENT CENTERS

Element	Sig xx	Sig yy	Sig zz	Sig xy	PWP	F
1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
2	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
3	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
4	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
5	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
6	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
7	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
8	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
9	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
10	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
11	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
12	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
13	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
14	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
15	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
16	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
17	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
18	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
19	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
20	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
21	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
22	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
23	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
24	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000

SUMMARY OF LOADING AND CONSTRUCTION EVENTS FOR STEP 1

SUMMARY OF ACTIVITIES:

```
Total time step ..................   1.00000
Number of Increments .............         1
Total time at end of step ........   1.00000
Number of equations ..............        64
Stiffness size (bytes) ...........       751

Boundary pressures applied.........        4
```

*** SUMMARY OF ACTIVE ELEMENTS ***

Elem	Status
1	Soil No. 1 existing below water table
2	Soil No. 1 existing below water table
3	Soil No. 1 existing below water table
4	Soil No. 1 existing below water table
5	Soil No. 1 existing below water table
6	Soil No. 1 existing below water table
7	Soil No. 1 existing below water table
8	Soil No. 1 existing below water table
9	Soil No. 1 existing below water table
10	Soil No. 1 existing below water table
11	Soil No. 1 existing below water table
12	Soil No. 1 existing below water table
13	Soil No. 1 existing below water table
14	Soil No. 1 existing below water table
15	Soil No. 1 existing below water table

16	Soil No.	1	existing	below	water table
17	Soil No.	1	existing	below	water table
18	Soil No.	1	existing	below	water table
19	Soil No.	1	existing	below	water table
20	Soil No.	1	existing	below	water table
21	Soil No.	1	existing	below	water table
22	Soil No.	1	existing	below	water table
23	Soil No.	1	existing	below	water table
24	Soil No.	1	existing	below	water table

*** SUMMARY OF ACTIVE NODES ***

Node		Status	DOF-X	DOF-Y	DOF-PWP	FX	FY
1	Old	Permanent boundary	Fixed	Fixed	0	0.0000	0.0000
2	Old	Permanent boundary	1	Fixed	Fixed	0.0000	0.0000
3	Old	Permanent boundary	2	Fixed	Fixed	0.0000	0.0000
4	Old	Permanent boundary	3	Fixed	Fixed	0.0000	0.0000
5	Old	Permanent boundary	4	Fixed	0	0.0000	0.0000
6	Old	Surface boundary	5	6	Fixed	0.0000	0.0000
7	Old	Interior	7	8	Fixed	0.0000	0.0000
8	Old	Interior	9	10	Fixed	0.0000	0.0000
9	Old	Interior	11	12	Fixed	0.0000	0.0000
10	Old	Surface boundary	13	14	Fixed	0.0000	0.0000
11	Old	Surface boundary	15	16	Fixed	0.0000	0.0000
12	Old	Interior	17	18	Fixed	0.0000	0.0000
13	Old	Interior	19	20	Fixed	0.0000	0.0000
14	Old	Interior	21	22	Fixed	0.0000	0.0000
15	Old	Surface boundary	23	24	Fixed	0.0000	0.0000
16	Old	Surface boundary	25	26	Fixed	0.0000	0.0000
17	Old	Interior	27	28	Fixed	0.0000	0.0000
18	Old	Interior	29	30	Fixed	0.0000	0.0000
19	Old	Interior	31	32	Fixed	0.0000	0.0000
20	Old	Surface boundary	33	34	Fixed	0.0000	0.0000
21	Old	Surface boundary	35	36	Fixed	0.0000	0.0000
22	Old	Interior	37	38	Fixed	0.0000	0.0000
23	Old	Interior	39	40	Fixed	0.0000	0.0000
24	Old	Interior	41	42	Fixed	0.0000	0.0000
25	Old	Surface boundary	43	44	Fixed	0.0000	0.0000
26	Old	Surface boundary	45	46	Fixed	0.0000	0.0000
27	Old	Interior	47	48	Fixed	0.0000	0.0000
28	Old	Interior	49	50	Fixed	0.0000	0.0000
29	Old	Interior	51	52	Fixed	0.0000	0.0000
30	Old	Surface boundary	53	54	Fixed	0.0000	0.0000
31	Old	Surface boundary	55	56	Fixed	0.0000	-5.0000
32	Old	Surface boundary	57	58	Fixed	0.0000	-10.0000
33	Old	Surface boundary	59	60	Fixed	0.0000	-10.0000
34	Old	Surface boundary	61	62	Fixed	0.0000	-10.0000
35	Old	Surface boundary	63	64	Fixed	0.0000	-5.0000

* = Side node with no pore pressure dof.

RESIDUAL = 0.562E-14 after 1 iterations.

DISPLACEMENTS AND PORE PRESSURES FOR T = 1.000

Node	Disp x	Disp y	PWP
1	0.0000	0.0000	0.0000
2	0.4444E-02	0.0000	0.0000
3	0.8888E-02	0.0000	0.0000
4	0.1333E-01	0.0000	0.0000
5	0.1778E-01	0.0000	0.0000
6	-0.1585E-15	-0.1778E-01	0.0000
7	0.4444E-02	-0.1778E-01	0.0000
8	0.8888E-02	-0.1778E-01	0.0000

9	0.1333E-01	-0.1778E-01	0.0000
10	0.1778E-01	-0.1778E-01	0.0000
11	-0.3413E-15	-0.3556E-01	0.0000
12	0.4444E-02	-0.3556E-01	0.0000
13	0.8888E-02	-0.3556E-01	0.0000
14	0.1333E-01	-0.3556E-01	0.0000
15	0.1778E-01	-0.3556E-01	0.0000
16	-0.5724E-15	-0.5333E-01	0.0000
17	0.4444E-02	-0.5333E-01	0.0000
18	0.8888E-02	-0.5333E-01	0.0000
19	0.1333E-01	-0.5333E-01	0.0000
20	0.1778E-01	-0.5333E-01	0.0000
21	-0.8616E-15	-0.7111E-01	0.0000
22	0.4444E-02	-0.7111E-01	0.0000
23	0.8888E-02	-0.7111E-01	0.0000
24	0.1333E-01	-0.7111E-01	0.0000
25	0.1778E-01	-0.7111E-01	0.0000
26	-0.1180E-14	-0.8889E-01	0.0000
27	0.4444E-02	-0.8889E-01	0.0000
28	0.8888E-02	-0.8889E-01	0.0000
29	0.1333E-01	-0.8889E-01	0.0000
30	0.1778E-01	-0.8889E-01	0.0000
31	-0.1513E-14	-0.1067	0.0000
32	0.4444E-02	-0.1067	0.0000
33	0.8888E-02	-0.1067	0.0000
34	0.1333E-01	-0.1067	0.0000
35	0.1778E-01	-0.1067	0.0000

SUMMARY OF STRESSES AT ELEMENT CENTERS

Element	Sig xx	Sig yy	Sig zz	Sig xy	PWP	F
1	0.1871E-13	10.00	3.333	0.1283E-13	0.0000	0.0000
2	0.1040E-13	10.00	3.333	0.1187E-13	0.0000	0.0000
3	0.2404E-14	10.00	3.333	0.1033E-13	0.0000	0.0000
4	-.5244E-15	10.00	3.333	0.4568E-14	0.0000	0.0000
5	0.4252E-15	10.00	3.333	0.3861E-14	0.0000	0.0000
6	0.3168E-14	10.00	3.333	0.1073E-13	0.0000	0.0000
7	0.7948E-15	10.00	3.333	0.1122E-13	0.0000	0.0000
8	0.2863E-14	10.00	3.333	0.4582E-14	0.0000	0.0000
9	0.7093E-15	10.00	3.333	0.9400E-15	0.0000	0.0000
10	0.2644E-14	10.00	3.333	0.1095E-13	0.0000	0.0000
11	0.7211E-14	10.00	3.333	0.3062E-14	0.0000	0.0000
12	-.1617E-14	10.00	3.333	-0.5367E-14	0.0000	0.0000
13	0.5024E-14	10.00	3.333	0.4163E-15	0.0000	0.0000
14	-.1243E-14	10.00	3.333	0.8928E-14	0.0000	0.0000
15	0.8423E-15	10.00	3.333	0.8603E-14	0.0000	0.0000
16	0.2302E-14	10.00	3.333	-0.4861E-14	0.0000	0.0000
17	0.5263E-14	10.00	3.333	0.7981E-15	0.0000	0.0000
18	0.1227E-13	10.00	3.333	-0.5277E-14	0.0000	0.0000
19	0.5681E-14	10.00	3.333	0.3215E-14	0.0000	0.0000
20	-.4185E-15	10.00	3.333	0.4823E-15	0.0000	0.0000
21	-.6206E-14	10.00	3.333	0.2131E-14	0.0000	0.0000
22	-.8459E-14	10.00	3.333	-0.2852E-14	0.0000	0.0000
23	-.1210E-13	10.00	3.333	0.5198E-14	0.0000	0.0000
24	0.2588E-14	10.00	3.333	0.4489E-14	0.0000	0.0000

Comments on the output file:

a. Note that all input data is summarized in the output.

b. Note that all initial displacements are zero. Initial stresses are zero only because the material was weightless.

c. The Y-displacement at nodes 31 through 35 were each -0.1067 in.

An edited version of the input file, ate_1.dat, for the eight-node problem follows. **Only the lines that are different** from the four-node data file are shown.

Eight-Node Elements

```
BASIC MESH SERIES 8-node elements.

C   filename = ATE_1.DAT.

KW XELT
   1:6
.
KW ELEM
    1      1      1      3      13     11     2      8      12     6
    2      1      3      5      15     13     4      10     14     8
    3      1      11     13     23     21     12     18     22     16
    4      1      13     15     25     23     14     20     24     18
    5      1      21     23     33     31     22     28     32     26
    6      1      23     25     35     33     24     30     34     28
.
KW CSBL
  31 32 33 -10 -10 -10
  33 34 35 -10 -10 -10
```

Comments on the input file:

a. The header line (first line of the file) is appropriately named..

b. The number of preexisting elements is 6 rather than 24 for the case of the four-node elements.

c. The connectivity of the elements as shown in KW ELEM is much different than the four-node table.

d. The KW CSBL is different from the form used for the four-node elements. The boundary has midside nodes in the case of eight-node elements.

An edited version of the input file, sixs1.dat, for the six-node problem with the horizontal three-node sides follows. ***Only the lines that are different from the four-node data file are shown.***

Six-Node Elements

```
BASIC MESH SERIES 6-node elements (horizontal orientation)
.
C filename = SIXS1.DAT (3-node side horizontal)
.
KW XELT
   1:12
.
KW ELEM
   1      1      1      3      8      6      2      0      7      0
   2      1      3      5      10     8      4      0      9      0
   3      1      6      8      13     11     7      0      12     0
   4      1      8      10     15     13     9      0      14     0
   5      1      11     13     18     16     12     0      17     0
   6      1      13     15     20     18     14     0      19     0
   7      1      16     18     23     21     17     0      22     0
   8      1      18     20     25     23     19     0      24     0
   9      1      21     23     28     26     22     0      27     0
  10      1      23     25     30     28     24     0      29     0
  11      1      26     28     33     31     27     0      32     0
  12      1      28     30     35     33     29     0      34     0
.
.
KW CSBL
```

```
  31   32   33   -10 -10 -10
  33   34   35   -10 -10 -10
```

Comments on the input:

a. Header matches problem.

b. KW XELT declares 12 preexisting elements as opposed to 24 for the four-node and 6 for the eight-node.

c. KW ELEM configures the nodes into six-node solid elements with the three-node sides horizontal.

d. KW CSBL is formatted for three-node side boundary similar to eight-node element.

An edited version of the input file for the six-node problem with the vertical three-node sides follows. Only the lines that are different from the four-node data file are shown plus the CSBL lines are shown to illustrate the difference between the mesh in sixs1.dat from sixs2.dat.

```
BASIC MESH SERIES 6-node elements (vertical orientation)
.
C filename = SIXS2.DAT
.
KW XELT
   1:12
.
KW ELEM
   1     1     2     12    11     1     7     0     6     0
   2     1     3     13    12     2     8     0     7     0
   3     1     4     14    13     3     9     0     8     0
   4     1     5     15    14     4    10     0     9     0
   5     1    12     22    21    11    17     0    16     0
   6     1    13     23    22    12    18     0    17     0
   7     1    14     24    23    13    19     0    18     0
   8     1    15     25    24    14    20     0    19     0
   9     1    22     32    31    21    27     0    26     0
  10     1    23     33    32    22    28     0    27     0
  11     1    24     34    33    23    29     0    28     0
  12     1    25     35    34    24    30     0    29     0
.
KW CSBL
  31    0   32   -10 -10 -10
  32    0   33   -10 -10 -10
  33    0   34   -10 -10 -10
  34    0   35   -10 -10 -10
```

Comments on the output:

a. Header matches problem.

b. KW XELT declares 12 preexisting elements as opposed to 24 for the four-node and 6 for the eight-node.

c. KW ELEM configures the nodes into six-node solid elements with the three-node sides vertical.

d. KW CSBL is formatted for two-node side boundary similar to four-node element.

An edited version of the input file for the five-node problem that illustrates the use of a transition element follows. Only the lines that are different from the four-node data file are shown.

Five-Node Transition Elements

```
BASIC MESH SERIES  5-node transition elements
.
C  filename = TRANS1.DAT
C  example using zero unit weight and 10 psi boundary load along
top.
C  eight node element at the top, 5-node in the center,
C  and 4-node on the bottom..
.
KW XELT
   1:6
.
KW ELEM
   1      1      1      3     13     11      0      0      0     0
   2      1      3      5     15     13      0      0      0     0
   3      1     11     13     23     21      0      0     22     0
   4      1     13     15     25     23      0      0     24     0
   5      1     21     23     33     31     22     28     32    26
   6      1     23     25     35     33     24     30     34    28
.
.
KW CSBL
  31 32 33 -10 -10 -10
  33 34 35 -10 -10 -10
```

Comments on the input:

a. Header matches problem.

b. KW XELT declares 6 preexisting elements as opposed to 24 for the four-node.

c. KW ELEM configures the nodes into 2 four-node elements, 2 five-node elements, and 2 eight-node element.

d. KW CSBL is formatted for the eight-node elements that compose the top of the block.

Axisymmetric Problems

Three-dimensional problems that are symmetric about some axis of rotation can be solved as a two-dimensional problem. Such problems are called axisymmetric problems. The code contains solid elements that are structured to simulate axisymmetric elements. In the code, the axis of symmetry is the Y-axis. The basic mesh series nodes were used to generate a mesh for a circular column of material that is subjected to a uniform stress at the top.

Closed form solution

The problem is that of a rod in uniaxial compression. The shortening of the rod can be expressed as:

Δ = PL /AE where P is the applied loading,
A is the cross sectional area,
L is the length of the rod
E is the modulus of elasticity

Stress (σ) can be substituted for the term P/A.

In the problem, the rod is 12 in. long, the modulus is 1,000 psi, and the applied stress (σ) is 10 psi.

The displacement of the top of the rod is

$\Delta = \sigma L/E = 10\,(12)/1{,}000 = 0.12$ in ↓ (in the direction of the applied stress)

The element axisymmetric elements available are four corner solid element and eight corner solid elements. The other solid element types (3-, 5-, 6-, and 7-node elements) must not be used for axisymmetric problems. If these other elements are used, the problem will run, but the answers determined by the program are not correct.

Eight-Node Elements

The input file, ate_1ax.dat, for the eight-node case is provided below:

```
STUBBs Hand Mesh # 2  8-node elements
C  July 25, 1995
C  filename = ATE_1AX.DAT
C  example using zero unit weight and 10 psi boundary load along top
C  of specimen (axisymmetric)
KW UNIT
C  ATMOSPHERIC PRESSURE, GAMMA H2O, VALUE OF 1-g (3 required
parameters)
C     psi,                    pci,       units of g
      14.7                   0.0361       1.0
C
KW SPEC
C FLOW, DFORM, DRAIN, CHECK_MESH, CHECK_SCRIPT, AXIS (6)
    F      T      T         F           F          T
C
KW XELT
C  Element existing at the start of the problem
   1:6
KW SMPR
C Linear elastic parameters
C MAT#,   GAMMAT, RVOID,   GS,   RSAT (5)
    1      0.00    0.55   2.7    .80
C MODULUS, POISX, POISY,  XN,   XM,   KO (6)
    1000   .3333   .3333   1.0  0.4  0.5
C PERMX, PERMY, B-Value  (3)
  1.0E-2  1.0E-2  0.999
C van Genuchten parameters (2)
     0.006  2.3
C  End of material  properties
```

```
C
KW NODES
C    Scaling properties  X-factor, Y-factor
     1.0        1.0
C    Node  X-coord      Y-coord    X     Y     PWP  dof's
      1    0.00         0.00       1     1     0
      2    1.0          0.000      0     1     0
      3    2.0          0.000      0     1     0
      4    3.00         0.000      0     1     0
      5    4.00         0.000      0     1     0
      6    0.00         2.000      1     0     0
      7    1.000        2.000      0     0     0
      8    2.000        2.000      0     0     0
      9    3.000        2.000      0     0     0
     10    4.000        2.000      0     0     0
     11    0.000        4.000      1     0     0
     12    1.000        4.000      0     0     0
     13    2.000        4.000      0     0     0
     14    3.000        4.000      0     0     0
     15    4.000        4.000      0     0     0
     16    0.000        6.000      1     0     0
     17    1.000        6.000      0     0     0
     18    2.000        6.000      0     0     0
     19    3.000        6.000      0     0     0
     20    4.000        6.000      0     0     0
     21    0.000        8.000      0     0     0
     22    1.000        8.000      0     0     0
     23    2.000        8.000      0     0     0
     24    3.000        8.000      0     0     0
     25    4.000        8.000      0     0     0
     26    0.000       10.000      1     0     0
     27    1.000       10.000      0     0     0
     28    2.000       10.000      0     0     0
     29    3.000       10.000      0     0     0
     30    4.000       10.000      0     0     0
     31    0.000       12.000      0     0     0
     32    1.000       12.000      0     0     0
     33    2.000       12.000      0     0     0
     34    3.000       12.000      0     0     0
     35    4.000       12.000      0     0     0
C
KW ELEM
C   8-node connectivity table
C Ele#   Mat #      4 corner nodes             midside nodes
    1      1      1     3     13    11     2     8      12    6
    2      1      3     5     15    13     4    10      14    8
    3      1     11    13     23    21    12    18      22   16
    4      1     13    15     25    23    14    20      24   18
    5      1     21    23     33    31    22    28      32   26
    6      1     23    25     35    33    24    30      34   28
C
KW LFTC
C  Start of construction script
KW TIME
C # of substeps, duration
     1             .01
C  Tot = 0.01
KW CSBL
C  Distributed load
C  Nodes      Pressure
 31 32 33   -10 -10 -10
 33 34 35   -10 -10 -10
KW DONE
KW DONE
```

Comments on the input file:

a. Note that the logical variable for axisymmetry is set to true under KW SPEC.

b. Note that the X-displacement degree of freedom was set to fixed for the nodes on the X axis (X = 0 line). However, for axisymmetric

problems it is not necessary to fix the displacements on the axis of symmetry.

c. The KW CSPL can not be used for axisymmetric problems.

The results of the analyzes are shown in Table C5. The four-node element mesh and the eight-node element mesh yielded a solution for the displacement of the rod that was in agreement with the closed-form solution.

Table C5
Comparison of Solutions - Axisymmetric Problems

Hand solution	0.12 in ↓
Four-node element Mesh (fours1ax)	0.12 in ↓
Eight-node element mesh (ate 1ax)	0.12 in ↓

Other example problems

The format for the remaining examples is shown below.

descriptive title:	**SIMPLE DESCRIPTIVE TITLE (bold print)**
filename:	name used in fea_file
narrative description:	brief description of the problem, citation of source (if applicable)
mesh (figure):	figure showing nodes and elements
hand solution:	hand solution for specific displacements or stresses
input file:	a copy of the input file
output file:	selected portions of the output file
comments:	comparison of hand solution and STUBBS output

Beam problem

filename: BEAM_T10.DAT

narrative description: A cantilever beam with a load applied at the free end. The displacement at the free end and the shear stress at the fixed support are calculated.

mesh (figure): Figure showing nodes and elements, Figure C3

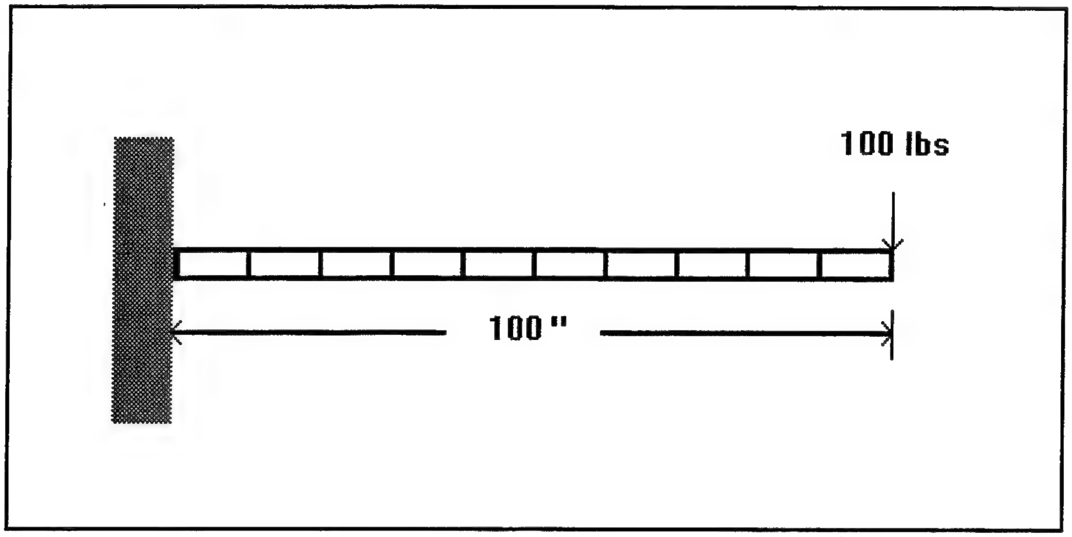


Figure C3. Cantilever beam

hand solution:

Given:

beam length (l) = 100 in., beam self weight (W) = 0 lbs
cross section area (A) = 2 sq. ft, modulus (E) = 1,000,000 psi
moment of inertia (I) = 10 in ^{4} , Poisson's Ratio = 0.25
modulus of rigidity (G) = $E/(2\ (1+\nu)) = 0.4 \times 10^6$ psi
applied loading at free end (P) = 100 lbs

Solution:

shear at support = P = 100 lbs
moment a support = Pl = 10,000 lbs-in
displacement at free end = $Pl^3 / 3EI + 1.2Pl/AG$

$(100)\ (100)^3 / (3)\ (10)\ (10^6)$	= 3.333 in ↓
$1.2\ (100)\ (100) / (2)\ (.4)\ (10^6)$	= 0.015 in ↓
total displacement	= 3.348 in ↓

input file: a copy of the input file

```
Cantilever Beam - ten beam elements
C   July 25, 1995
KW UNIT...Units conversion card
```

```
C  ATMOSPHERIC PRESSURE,  GAMMA H2O,  VALUE OF 1-g  (3 required
parameters)
C      psi,                  pci,        units of g   Residual
criteria
    14.7                    0.0361         1.0         1E-6
KW SPEC ...Defines general problem options
C  FLOW, DFORM, DRAIN, CHECK_MESH, CHECK_SCRIPT, AXIS
     F     T      T         F            F          F
KW NODE...Nodal data
C The first card of this group is the x-scale and y-scale factors.
C    Scaling parameters X-factor, Y-factor
    10.0     1.0
C  Node    X-coord      Y-coord    X   Y  PWP dof's
     1       0.0         0.0       1   1   0
     2       0.1         0.0       1   1   0
C
     3       0.0         5.0       0   0   0
     4       0.1         5.0       0   0   0
C
     5       0.0        10.0       0   0   0
     6       0.1        10.0       0   0   0
C
     7       0.0        15.0       0   0   0
     8       0.1        15.0       0   0   0
C
     9       0.0        20.0       0   0   0
    10       0.1        20.0       0   0   0
C
    11       0.0        25.0       0   0   0
    12       0.1        25.0       0   0   0
C
    13       0.0        30.0       0   0   0
    14       0.1        30.0       0   0   0
C
    15       0.0        35.0       0   0   0
    16       0.1        35.0       0   0   0
C
    17       0.0        40.0       0   0   0
    18       0.1        40.0       0   0   0
C
    19       0.0        45.0       0   0   0
    20       0.1        45.0       0   0   0
C
    21       0.0        50.0       0   0   0
    22       0.1        50.0       0   0   0
C
    23       0.0        55.0       0   0   0
    24       0.1        55.0       0   0   0
C
    25       0.0        60.0       0   0   0
    26       0.1        60.0       0   0   0
C
    27       0.0        65.0       0   0   0
    28       0.1        65.0       0   0   0
C
    29       0.0        70.0       0   0   0
    30       0.1        70.0       0   0   0
C
    31       0.0        75.0       0   0   0
    32       0.1        75.0       0   0   0
C
    33       0.0        80.0       0   0   0
    34       0.1        80.0       0   0   0
C
    35       0.0        85.0       0   0   0
    36       0.1        85.0       0   0   0
C
    37       0.0        90.0       0   0   0
    38       0.1        90.0       0   0   0
C
    39       0.0        95.0       0   0   0
    40       0.1        95.0       0   0   0
C
    41       0.0       100.0       0   0   0
    42       0.1       100.0       0   0   0
C
KW XBND...Preexisting elements
```

```
C
        1:10
C
KW BEND...Bending element connectivity data
C
C  Ele  Mat#     4 corner nodes          2 midside nodes
    1   1    2      6      5      1      4      3
    2   1    6     10      9      5      8      7
    3   1   10     14     13      9     12     11
    4   1   14     18     17     13     16     15
    5   1   18     22     21     17     20     19
    6   1   22     26     25     21     24     23
    7   1   26     30     29     25     28     27
    8   1   30     34     33     29     32     31
    9   1   34     38     37     33     36     35
   10   1   38     42     41     37     40     39
C
KW BMMP...Bending section properties
C Mat#     E     Poissons ratio   I          A
   1     1.D6   0.25             10.0000     1.0
C
KW LFTC...Construction script.
C Increment specification
C
C ---- Point load ------
KW TIME
  1     .05
C
KW CSPL
 41 50. 0.
 42 50. 0.
C
KW DONE
KW DONE
```

Comments on input file:

a. Note that a scale factor of 10 was used in the X-direction in KW NODE.

b. The duration of 0.05 in the KW TIME data is arbitrary since no time dependant behavior is involved.

Selected output from BEAM_T10.OUT

Shear at support = 100 lb; same as hand solution
Moment at support = 100,000 lb-in.; same as hand solution

Comments:

The displacement at the free end (output) is 3.358 in.; expected from hand solution is 3.348 in. The shear and moment (output) are in agreement with the hand solution.

Simple truss (bar elements)

filename:	trus1.dat
narrative description:	A five-member, simply-supported truss is subjected a concentrated load of 10 lbs applied

at the center joint on the top of the truss frame. The vertical displacement of the center of the bottom chord is determined. The truss is statically determinate and all the members have the same modulus and cross section.

mesh (figure): Figure C-4

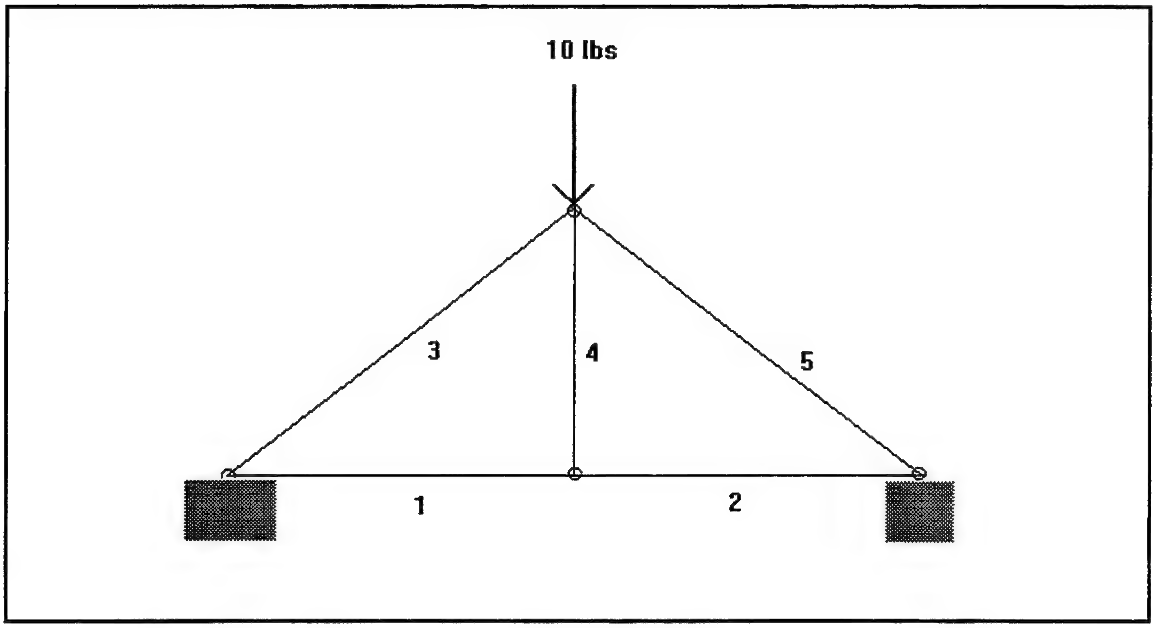


Figure C4. Simple truss

hand solution: hand solution for specific displacements or forces

The forces in the members can be solved by inspection. The vertical member No. 4 has zero load. The vertical component of members No. 3 and No. 5 is 5 lbs (compression). The members No. 3 and No. 5 are inclined at a slope of 1v:1h (45°). So, the horizontal component is 5 lbs (compression), and the axial force in the member is 5 $\sqrt{2}$ or 7.071 lbs (compression). The axial forces in members No. 1 and No. 2 is 5 lbs (tension). To find the vertical deflection at the midpoint of the bottom chord, a 1-lb dummy unit load is applied to the node. The forces in the members can be found by inspection and comparison with the real load solution. Member No. 4 carries 1-lb tension. Then, the other axial loads can be found from multiplying the loads due to the real 10-lb load by 0.1. Then, the loading due to the real and dummy-unit loads can be placed in Table 6.

Table C6
Truss Problem Solution Using Dummy-Unit-Load Method

Member	Length (in)	P,(lbs) real	e = PL/AE	p, (lbs) dummy	p e
1	10	5	0.005	0.5	0.0025
2	10	5	0.005	0.5	0.0025
3	14.14	-7.071	-0.01	-0.7071	0.007071
4	10	0	0	-1	0
5	14.14	-7.071	-0.01	-0.7071	0.007071

The vertical displacement at the midpoint of the bottom chord is $\sum$ pe = 0.019142 in. in the direction of the dummy-unit load (downward ↓).

input file:

```
STUBBs Hand Mesh 3-node elements - bar elements - SIMPLE TRUSS
C  July 25 ,1995
C  filename = trus1.dat
KW UNIT
C  ATMOSPHERIC PRESSURE,  GAMMA H2O,  VALUE OF 1-g  (3 required
parameters)
C     psi,               pci,       units of g     Residual
criterion
      14.7             0.0361        1.0            1E-6
C
KW SPEC
C FLOW, DFORM, DRAIN, CHECK_MESH, CHECK_SCRIPT, AXIS (6)
    F     T      T         F            F           F
C
KW BRMP
C Bar material properties
C  Note that stiffness (AE) is used, not simply modulus (E)
C MAT#, AEtensile, AEcompression
    1     1.D4         1.D4
C  End of material  properties
C
KW NODES
C   Scaling parameters X-factor, Y-factor
    1.0     1.0
C  Node    X-coord      Y-coord    X     Y     PWP dof's
    1      0.00         0.00      1     1     0
    2      5.00         0.000     0     0     0
    3     10.00         0.000     0     0     0
    4     15.00         0.000     0     0     0
    5     20.00         0.000     0     1     0
    6      5.00         5.000     0     0     0
    7     10.00         5.000     0     0     0
    8     15.00         5.000     0     0     0
    9     10.00        10.000     0     0     0
C
KW BREL
C Bar element connectivity table
C Ele#   Mat#      3  Nodes
    1     1     1     3     2
    2     1     3     5     4
    3     1     1     9     6
    4     1     3     9     7
    5     1     5     9     8
C
KW XBRE
C  Bars existing at the start of the problem
   1:5
C
```

```
KW LFTC
C  Start of construction script
KW TIME
C # of substeps, duration
       1           1.
C ---- Point load ------
KW CSPL
C  Node #  X-force  Y-force
      9      0.0    -10.0
KW DONE
KW DONE
```

Comments on input file:

Note that KW XBRE was positioned after the KW BREL. If the positions were reversed, the program would appear to run, but no forces and displacement would be calculated.

output file: selected portions of the output file

The displacement at the center of the truss was 0.01914 in ↓ and agrees with the hand solution. The forces in the members are also in agreement with the hand solution.

Table C7
Forces in Truss Members

Member	Hand Solution (lbs)	Output File (lbs)
1	5.0	5.0
2	5.0	5.0
3	-7.1	-7.1
4	0.0	0.19 E-13
5	-7.1	-7.1

Comments on the solution:

The force in member 4 determined by the program is essentially zero, in agreement with the hand solution.

Complex truss

filename: roark1.dat

narrative description: A cantilever truss with six members is subjected to three concentrated loads. The displacement of the free end truss is determined. The truss is statically determinate and the members have the same modulus but vary in cross section.

mesh (figure)   Figure C-5

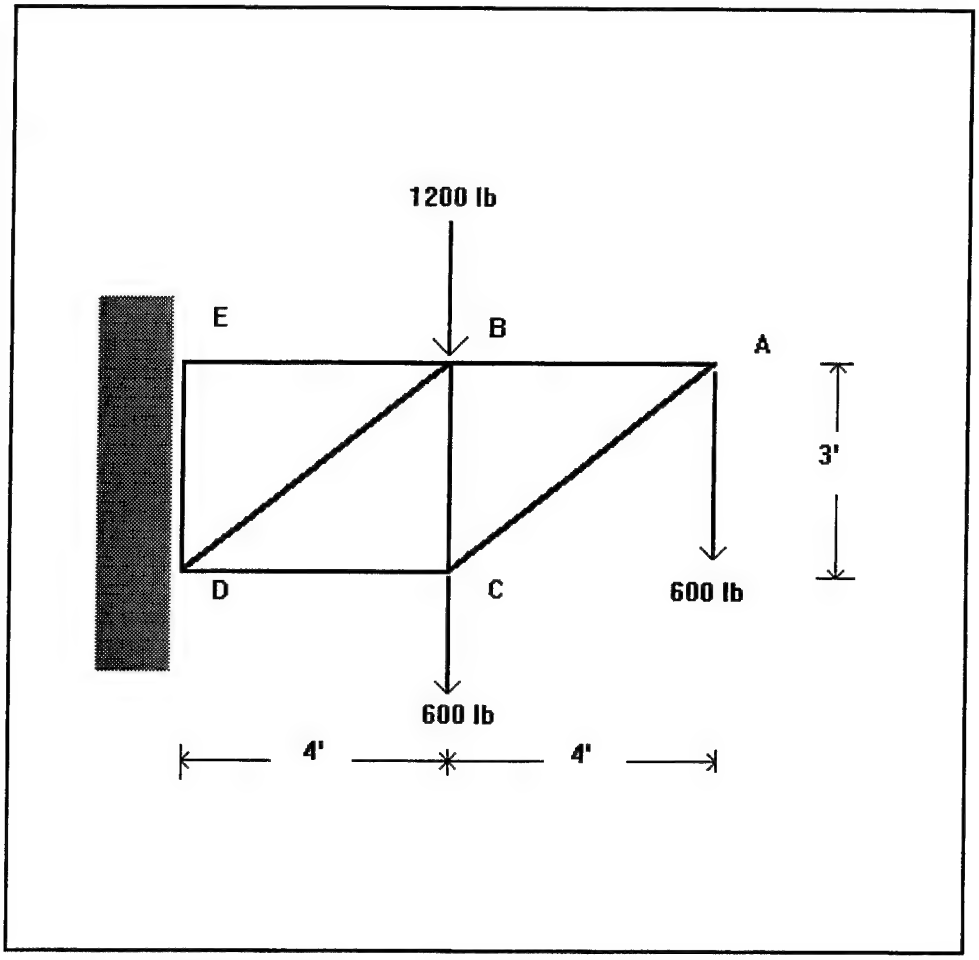


Figure C5. Complex truss

hand solution   Solution from Roark's Formulas for Stress and Strain 6th Ed. (1989)

Deflection at A X displacement = 0.0317 → Y displacement = 0.1769↓

input file: roark1.dat

```
STUBBs Roark page 83 truss 3-node elements - bar elements - SIMPLE
TRUSS
C  July 24, 1995
C  filename = ROARK1.DAT
KW UNIT
C  ATMOSPHERIC PRESSURE,  GAMMA H2O,  VALUE OF 1-g (3)
C      psi,                 pci,        units of g
       14.7               0.0361           1.0
C
KW SPEC
C FLOW, DFORM, DRAIN, CHECK_MESH, CHECK_SCRIPT, AXIS (6)
    F     T      T          F            F          F
C
```

```
KW BRMP
C Bar material properties
C  Note that stiffness (AE) is used, not simply modulus (E)
C MAT#,    AEten,      AEcom
   1     2358600     2358600
   2     4392000     4392000
   3    12426000    12426000
   4     9954000     9954000
C  End of material properties
C
KW NODES
C   Scaling parameters X-factor, Y-factor
                           12.0      12.0
C  Node    X-coord      Y-coord  X     Y      PWP dof's
    1      0.00         0.00     1     1      0
    2      2.0          0.00     0     0      0
    3      4.0          0.00     0     0      0
    4      0.00         1.50     0     0      0
    5      2.00         1.50     0     0      0
    6      4.00         1.50     0     0      0
    7      6.00         1.50     0     0      0
    8      0.00         3.00     1     1      0
    9      2.00         3.00     0     0      0
   10      4.00         3.00     0     0      0
   11      6.00         3.00     0     0      0
   12      8.00         3.00     0     0      0
C
KW BREL
C   Bar connectivity table
C  Bar #   Mat #       3 nodes
    1       1       1     3     2
    2       4       1    10     5
    3       3       8    10     9
    4       2       3    10     6
    5       1       3    12     7
    6       1      10    12    11
C
KW XBRE
C  Bars existing at the start of the problem
   1:6
C
KW LFTC
C  Start of construction script
KW TIME
C # of substeps, duration
      1             1.
C ---- Point load ------
KW CSPL
  3  0     -600.0
 10  0    -1200.0
 12  0     -600.0
KW DONE
KW DONE
```

Comments on input file:

a. Four material types were used to accommodate changes in cross-section.

b. A scale factor of 12 for X- and Y- was used to allow coordinates input in ft and output in inches.

Selected output from roark1.out:

The results calculated for the forces in the members agreed with the hand calculation as shown in the Table 8. Details of the Truss.

Table C8
Details of the Truss

Member	Area, A, in^2	Length, l, in	P, lb
AB	0.07862	48	+ 800
AC	0.07862	60	- 1000
BC	0.1464	36	+1200
BE	0.4142	48	+4000
BD	0.3318	60	- 4000
CD	0.07862	48	- 800

Table C9
Comparison of Truss Displacements

Displacements at Point A	Results from Output	Hand Calculation
X - displacement	0.03173 in →	0.0317 in→
Y - displacement	-0.1770 in ↓	-0.1769in ↓

Slip-separation solutions

filename: Reed2b.dat

Two solid elements are separated by a slip-separation element. The concept for this problem was found in a report by Mosher (1992) [U.S. Army Waterways Experiment Station Technical Report ITL-92-1, R. L. Mosher Three-Dimensional Finite Element Analysis of Sheet-Pile Cellular Cofferdams.] Mosher, in turn, used a problem presented by Desai et al. (1984) [C. S. Desai, M. M. Zaman, J. G. Lighter, and H. J. Siriwardane. Thin-Layer element for interfaces and joints. *International Journal for Numerical and Analytical Methods in Geomechanics*, Vol. 8, pp 19-43., 1984]. The system of solid and slip elements initially forms one continuous block. The two solid elements have identical material properties. When each block is subjected to a uniform vertical load (2000 psi on left-hand block, 1000 psi on right-hand block), the blocks deform as two independent units linked by the slip element. The interface properties of the slip-separation element determine the degree of independence of the two solid blocks. When the interface is stiff (low compliance), the blocks act as a continuous solid. When the interface is soft (high compliance), the blocks act independently. Intermediate values of stiffness (compliance) produce moderate linkage of the deformation of the two solid blocks. The condition when the interface is infinitely stiff (non-compliant) was modeled by removing the slip element and joining the two solid elements. The input file for the infinitely stiff (noncompliance) interface was named reed2a.dat. In fact, the infinitely stiff case included no slip element; the two blocks were fixed together.

mesh (figure): Figure C-6

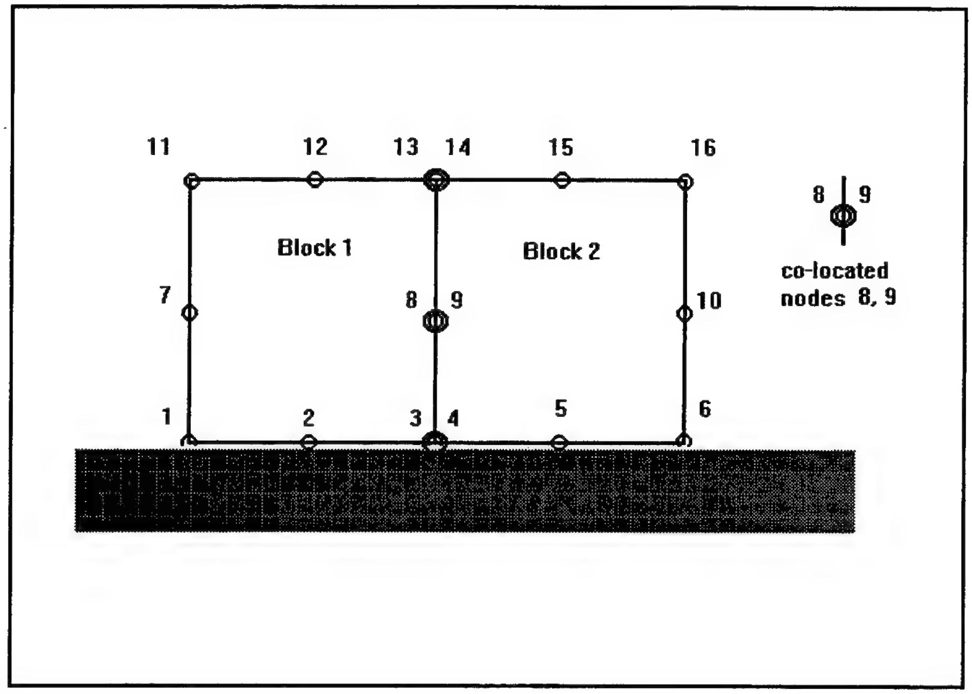


Figure C6. Slipping blocks

hand solution: hand solution for specific displacements.

For the uniform vertical load and no horizontal stress, the strain is

$$\epsilon_y = \frac{\sigma_y}{E}(1-\nu^2)$$

The displacement Δ is $\epsilon_y H$, where H is the height of the block. So,

Δ = 1000/10000 (1-0.3 ^{2}) x 10 or
0.91 in for a loading of 1000 psi, and 1.82 in for a loading of 2000 psi.

input file : a copy of the input file

```
SLIP ELEMENT TEST - "variation of example 2 from reed mosher's
dissertation"
C  July 25, 1995
C  filename = REED2B.DAT
C  two-element problem with slip element in between
C  note blocks are weightless - units are in lbs and inches
KW SPEC ...Defines general problem options
C  FLOW, DFORM, DRAIN, CHECK_MESH, CHECK_SCRIPT, AXIS
     F     T      T        F           F         F
KW NODE...Nodal data
```

```
C   Scaling parameters X-factor, Y-factor
    1.0      1.0
C  Node    X-coord Y-coord  X    Y   PWP dof's
    1      -10.0    0.0     0    1    0
    2       -5.0    0.0     0    1    0
    3        0.0    0.0     1    1    0
    4        0.0    0.0     0    1    0
    5        5.0    0.0     0    1    0
    6       10.0    0.0     0    1    0
    7      -10.0    5.0     0    0    0
    8        0.0    5.0     0    0    0
    9        0.0    5.0     0    0    0
   10       10.0    5.0     0    0    0
   11      -10.0   10.0     0    0    0
   12       -5.0   10.0     0    0    0
   13        0.0   10.0     0    0    0
   14        0.0   10.0     0    0    0
   15        5.0   10.0     0    0    0
   16       10.0   10.0     0    0    0
C
KW ELEM .... soil elements
C    4-node connectivity table
C Ele#  Mat #    corner nodes   mid-side nodes
   1     1     1   3   13  11    2   8  12   7
   2     2     4   6   16  14    5  10  15   9
C
KW SLIP
C     6-node slip connectivity table
C Ele#  Mat #    corner nodes    mid-side nodes
   1     3        4  14    13   3    9   8   0   0
C
KW UNIT ..... Units conversion data
C  units are in inches, and lbs
      14.7     0.0361   1.0
C
KW SMPR .... soil material properties
C  linear elastic material properties
C MAT#,   GAMMAT, RVOID, GS, RSAT (5)
   1      0.0    1.0    2.7   1.0
C MODULUS,  POISX,  POISY,    XN,   XM,    Ko (6)
  1.0E4      0.30    0.30     1.0   0.4    0.999
C PERMX,   PERMY, B-Value   (3)
   0.012  0.012     0.98
C  van Genuchten parameters (2)
    0.006  2.3
C MAT#,   GAMMAT, RVOID, GS, RSAT (5)
   2    0.0    1.0    2.7    1.0
C MODULUS,  POISX,  POISY,    XN,   XM,   Ko (6)
  1.0E4      0.30    0.30     1.0  0.4    0.999
C PERMX, PERMY, B-Value  (3)
    0.012     0.012     0.98
C  van Genuchten parameters (2)
    0.006  2.3
C
C
KW SLMP
C    MAT #     Tension    Friction        Cohesion     compliances
C                                                        shear   normal
     3          0.          0.0           3000.        1.0D-5  1.D-8
C
KW XELT
   1:2
C
KW XSLP
   1:1
C
KW LFTC
C  Start of construction script
KW TIME
C # of substeps, duration
   1   1.
C
KW CSBL  ..... apply normal load
C Distributed load
C     Nodes             Pressure
    11  12  13   -200.   -200.   -200.
```

```
    14   15   16    -200.     -200.     -200.
KW DONE
KW DONE
```

Comments on input file:

a. Two identical material types were used.

b. The bottom center node (node 3) was fixed in both the X- and Y-directions.

c. The input file shows that .....

output file: None shown. - the problem was run several times with different values for compliance for the slip element information. The displacement for the top right-hand node of block 1 (node 13) and the top left-hand node of block 2 (node 14) were compared and the results presented in the table below.

comments: comparison of hand solution and STUBBS output

Note that in Table C10 the displacements are in inches and are negative in the output file. The negative sign indicates downward movement.

Table C10
Comparison of Block Vertical Displacements of Adjacent Block Corners

Compliance / Fixity of Slip Element	Node 13 Block 1	Node14 Block 2	Vertical Separation
Infinity / fixed	1.365	1.365	0.000
1E -05	1.368	1.362	0.006
0.001	1.498	1.232	0.266
0.01	1.714	1.016	0.698
0.1	1.806	0.924	0.882
1	1.819	0.911	0.908
10	1.820	0.910	0.910
Free	1.820	0.910	0.910
Hand solution	1.820	0.910	0.910

Flow Problems

STUBBS can solve both transient and steady state flow problems. Two example problems showing steady-state flow are presented. Both of the problems used the basic mesh series (BMS) configuration with 24 four-node elements.

The problems have identical boundary conditions that are similar to a constant head test. The gradient across the specimen is identical in both tests. Both problems involve two soils with vastly different permeabilities. For ease of hand checking values, a fluid unit weight of 50 lbs per cu ft was used. Likewise, the dimensions of the specimen are not intended to model any actual test conditions.

One soil has relatively low permeability and the other material has a permeability an order of magnitude higher. In the discussion that follows the soils are called the fast soil (higher permeability) and the slow soil (lower permeability). In one problem, the soils are layered in a column and the flow path passes through one soil and then the other. In the other problem, the soils are side by side in the column. The dimensions and heads common to both problems are listed in Table C11.

Table C11
Dimension and Parameters Used in the Flow Problems

Specimen length	120 ft
Width	4 ft
Head difference	6 ft
Ratio of permeabilities (high/low)	10
Fluid unit weight	50 lbs/ cu ft
Gradient across specimen	0.05

The distribution of head is hydrostatic at the beginning of each test. Then, the head at the top of the sample is reduced to create an upward gradient. The expected flow velocity in a homogeneous sample of each soil subjected to the test gradient can be calculated from Darcy's Law ($v = ki$)

low permeability soil $k = 0.01$ ft/min $I = 0.05$ $v = 0.0005$ ft/min
(slow soil)

high permeability soil $k = 0.1$ ft/min $I = 0.05$ $v = 0.005$ ft/min
(fast soil)

In the test where the soil specimen is composed of a 60 ft layer of each soil. It is expected that almost all of the head loss will occur within the low

permeability material. For this case where the soils have then same thickness, the ratio of the head loss in the low permeability soil to the head loss in the higher permeability soil is the same as the ratio of the permeability of the high permeability soil to the permeability of the low permeability soil. Because the ratio of permeabilities is 10, the head loss through the low permeability soil should be 10 times the head loss through the higher permeability soil. So, 10/11 of the head loss through the specimen should occur in the low permeability soil and the remaining 1/11 of the head loss should take place in the higher permeability soil. The velocity through the two-soil-specimen will fall somewhere between the two values calculated for each single-soil specimen.

A summary of expected head loss, gradients, and velocities is shown in Table C12.

Table C12
Expected Values for Head Loss, Gradient, and Velocity

Flow Across Layers	Head Loss	Gradient	Velocity
Specimen	6 ft	0.05	9.09E-4
High permeability soil	0.55 ft	0.01	9.09E-4
Low permeability soil	5.45 ft	0.09	9.09E-4

The input file FLOW1.DAT is shown below:

```
BASIC MESH SERIES   24 four-node elements - two permeabilities
C  July 25, 1995
C  filename = FLOW1.DAT
C  Serial flow through material 2 then material 1
KW UNIT
C  ATMOSPHERIC PRESSURE,  GAMMA H2O,  VALUE OF 1-g  (3)
C     psf,                  pcf,        units of g
    2116.8                 50.0          1.0
C
KW SPEC
C FLOW, DFORM, DRAIN, CHECK_MESH, CHECK_SCRIPT, AXIS (6)
   T      F      F         F            F          F
C
KW XELT
C  solid element existing at the start of the problem
   1:24
KW SMPR
C  linear elastic material properties
C  *************  FAST SOIL - HIGH PERMEABILITY  *************
C MAT#,  GAMMAT, RVOID,   GS, RSAT (5)
   1     100.00   0.85    2.7   0.80
C MODULUS, POISX, POISY, XN,  XM,  Ko (6)
   1000   .3333  .3333  1.0  0.4  0.5
C PERMX, PERMY, B-Value  (3)
   0.1    0.1    0.999
C van Genuchten parameters (2)
    0.006    2.3
C **************  SLOW SOIL - LOW PERMEABILITY  **************
C MAT#,  GAMMAT, RVOID,    GS, RSAT (5)
   2     100.0    0.85    2.7  0.80
C MODULUS, POISX, POISY, XN,  XM,  Ko (6)
   1000   .3333   .3333  1.0  0.4  0.5
C PERMX, PERMY, B-Value  (3)
   0.01  0.01  0.999
```

```
C van Genuchten parameters (2)
    0.006     2.3
C
KW NODE
C   Scaling parameters X-factor, Y-factor
    1.0      10.0
C  Node     X-coord        Y-coord     X       Y      PWP dof's
    1       0.00           0.00        0       1       1
    2       1.0            0.000       0       1       1
    3       2.0            0.000       1       1       1
    4       3.00           0.000       0       1       1
    5       4.00           0.000       0       1       1
    6       0.00           2.000       0       0       0
    7       1.000          2.000       0       0       0
    8       2.000          2.000       0       0       0
    9       3.000          2.000       0       0       0
   10       4.000          2.000       0       0       0
   11       0.000          4.000       0       0       0
   12       1.000          4.000       0       0       0
   13       2.000          4.000       0       0       0
   14       3.000          4.000       0       0       0
   15       4.000          4.000       0       0       0
   16       0.000          6.000       0       0       0
   17       1.000          6.000       0       0       0
   18       2.000          6.000       0       0       0
   19       3.000          6.000       0       0       0
   20       4.000          6.000       0       0       0
   21       0.000          8.000       0       0       0
   22       1.000          8.000       0       0       0
   23       2.000          8.000       0       0       0
   24       3.000          8.000       0       0       0
   25       4.000          8.000       0       0       0
   26       0.000         10.000       0       0       0
   27       1.000         10.000       0       0       0
   28       2.000         10.000       0       0       0
   29       3.000         10.000       0       0       0
   30       4.000         10.000       0       0       0
   31       0.000         12.000       0       0       1
   32       1.000         12.000       0       0       1
   33       2.000         12.000       0       0       1
   34       3.000         12.000       0       0       1
   35       4.000         12.000       0       0       1
C
KW ELEM
C     4-node connectivity table
C Ele#  Mat #      4 corner nodes              midside nodes
C  Elements #1 thru #12 material #1 fast permeability
    1      1      1     2     7     6     0     0     0     0
    2      1      2     3     8     7     0     0     0     0
    3      1      3     4     9     8     0     0     0     0
    4      1      4     5    10     9     0     0     0     0
    5      1      6     7    12    11     0     0     0     0
    6      1      7     8    13    12     0     0     0     0
    7      1      8     9    14    13     0     0     0     0
    8      1      9    10    15    14     0     0     0     0
    9      1     11    12    17    16     0     0     0     0
   10      1     12    13    18    17     0     0     0     0
   11      1     13    14    19    18     0     0     0     0
   12      1     14    15    20    19     0     0     0     0
C  Elements #13 thru #24 material #2 slow permeability
   13      2     16    17    22    21     0     0     0     0
   14      2     17    18    23    22     0     0     0     0
   15      2     18    19    24    23     0     0     0     0
   16      2     19    20    25    24     0     0     0     0
   17      2     21    22    27    26     0     0     0     0
   18      2     22    23    28    27     0     0     0     0
   19      2     23    24    29    28     0     0     0     0
   20      2     24    25    30    29     0     0     0     0
   21      2     26    27    32    31     0     0     0     0
   22      2     27    28    33    32     0     0     0     0
   23      2     28    29    34    33     0     0     0     0
   24      2     29    30    35    34     0     0     0     0
C
KW IDPP
C  Initial node data
```

```
C    NODE        X-DISPL    Y-DISPL    PWP
      1          0.0        0.0        6300
      2          0.0        0.0        6300
      3          0.0        0.0        6300
      4          0.0        0.0        6300
      5          0.0        0.0        6300
      6          0.0        0.0        5300
      7          0.0        0.0        5300
      8          0.0        0.0        5300
      9          0.0        0.0        5300
      10         0.0        0.0        5300
      11         0.0        0.0        4300
      12         0.0        0.0        4300
      13         0.0        0.0        4300
      14         0.0        0.0        4300
      15         0.0        0.0        4300
      16         0.0        0.0        3300
      17         0.0        0.0        3300
      18         0.0        0.0        3300
      19         0.0        0.0        3300
      20         0.0        0.0        3300
      21         0.0        0.0        2300
      22         0.0        0.0        2300
      23         0.0        0.0        2300
      24         0.0        0.0        2300
      25         0.0        0.0        2300
      26         0.0        0.0        1300
      27         0.0        0.0        1300
      28         0.0        0.0        1300
      29         0.0        0.0        1300
      30         0.0        0.0        1300
      31         0.0        0.0         300
      32         0.0        0.0         300
      33         0.0        0.0         300
      34         0.0        0.0         300
      35         0.0        0.0         300
C
KW LFTC
C  Start of construction script
KW TIME
C # of substeps, duration
   1   10.
KW CSBC
C   Boundary conditions
C   NODE  X-DISPL    Y-DISPL    PWP-CODE  X-DISPL    Y-DISPL      PWR
    31         0          0          1    0.0        0.0           0.0
    32         0          0          1    0.0        0.0           0.0
    33         0          0          1    0.0        0.0           0.0
    34         0          0          1    0.0        0.0           0.0
    35         0          0          1    0.0        0.0           0.0
KW DONE
KW TIME
   10    50
KW DONE
KW TIME
   1   10
KW CSEC
C  Change material type -  Make all of specimen material 2
  1     2
  2     2
  3     2
  4     2
  5     2
  6     2
  7     2
  8     2
  9     2
 10     2
 11     2
 12     2
KW DONE
KW TIME
   10    50
KW DONE
KW DONE
```

Notice that sufficient time-steps were used to take the problem to a steady-state solution. The number of time steps and their duration was determined by trial and error. A comparison of expected and calculated quantities is shown in Table C13. The expected values in the table were rounded to the same number of digits.

Table C13
Comparison of Expected Values and STUBBS Output

FLOW1. OUT	Expected Values	Stubbs Output
Velocity	9.09E-4 ft/min	9.09E-4 ft/min
Head loss in slow soil	5.5 ft	5.5 ft
Head loss in fast soil	0.5 ft	0.5 ft

as shown in the output file. The output file in pure flow problems produces velocity and total head data for each node. A section of the output file FLOW1.OUT is shown below. The data for the first time-step (hydrostatic conditions) and the last time-step (steady-state flow) are shown.

```
 -   Flow Velocities and Pressures for T =    1.0000 -

Node        Vel x          Vel y           PWP     Total Head

 1       -0.2492E-15    -0.5633E-16     6300.       126.0
 2       -0.2492E-15    -0.2077E-16     6300.       126.0
 3       -0.2492E-15    -0.4131E-16     6300.       126.0
 4        0.3363E-15    -0.6404E-16     6300.       126.0
 5        0.9219E-15    -0.7616E-16     6300.       126.0
 6       -0.4548E-16    -0.5633E-16     5300.       126.0
 7       -0.4019E-32    -0.2077E-16     5300.       126.0
 8       -0.6544E-32    -0.5219E-16     5300.       126.0
 9        0.2535E-15    -0.8061E-16     5300.       126.0
10        0.6768E-16    -0.8753E-16     5300.       126.0
11       -0.2531E-16    -0.5633E-16     4300.       126.0
12       -0.4804E-16    -0.2077E-16     4300.       126.0
13       -0.4804E-16    -0.5219E-16     4300.       126.0
14       -0.6124E-16    -0.8061E-16     4300.       126.0
15       -0.2639E-15    -0.8753E-16     4300.       126.0
16        0.3452E-16    -0.2919E-16     3300.       126.0
17       -0.4279E-16    -0.1077E-16     3300.       126.0
18       -0.4279E-16    -0.2696E-16     3300.       126.0
19       -0.3845E-16    -0.4169E-16     3300.       126.0
20        0.2067E-15    -0.4994E-16     3300.       126.0
21        0.3395E-16    -0.3840E-17     2300.       126.0
22        0.1575E-16    -0.1420E-17     2300.       126.0
23        0.1575E-16    -0.3470E-17     2300.       126.0
24        0.2733E-16    -0.5410E-17     2300.       126.0
25        0.1845E-16    -0.1055E-16     2300.       126.0
26        0.4374E-16    -0.5633E-17     1300.       126.0
27        0.2101E-16    -0.2077E-17     1300.       126.0
28        0.2101E-16    -0.5219E-17     1300.       126.0
29        0.1159E-16    -0.8061E-17     1300.       126.0
30       -0.1374E-16    -0.8753E-17     1300.       126.0
31        0.7321E-16    -0.5633E-17      300.0      126.0
32        0.4593E-16    -0.2077E-17      300.0      126.0
33        0.4593E-16    -0.6308E-17      300.0      126.0
34        0.9286E-17    -0.9718E-17      300.0      126.0
35       -0.3645E-16    -0.9890E-17      300.0      126.0
```

FLOW VELOCITIES AND PRESSURES FOR T = 81.00

Node	Vel x	Vel y	PWP	Total Head
1	-0.5698E-15	0.9091E-03	6300.	126.0
2	-0.3038E-15	0.9091E-03	6300.	126.0
3	-0.8243E-16	0.9091E-03	6300.	126.0
4	0.1716E-15	0.9091E-03	6300.	126.0
5	0.4702E-15	0.9091E-03	6300.	126.0
6	0.2228E-14	0.9091E-03	5291.	125.8
7	0.1108E-14	0.9091E-03	5291.	125.8
8	-0.1421E-15	0.9091E-03	5291.	125.8
9	-0.8274E-15	0.9091E-03	5291.	125.8
10	-0.1473E-14	0.9091E-03	5291.	125.8
11	0.8973E-16	0.9091E-03	4282.	125.6
12	-0.1326E-14	0.9091E-03	4282.	125.6
13	-0.3145E-14	0.9091E-03	4282.	125.6
14	-0.2335E-14	0.9091E-03	4282.	125.6
15	-0.1289E-14	0.9091E-03	4282.	125.6
16	-0.7926E-15	0.9091E-03	3273.	125.5
17	-0.1190E-14	0.9091E-03	3273.	125.5
18	-0.1900E-14	0.9091E-03	3273.	125.5
19	-0.1277E-14	0.9091E-03	3273.	125.5
20	-0.4158E-15	0.9091E-03	3273.	125.5
21	-0.6642E-16	0.9091E-03	2182.	123.6
22	-0.2016E-15	0.9091E-03	2182.	123.6
23	-0.1789E-15	0.9091E-03	2182.	123.6
24	-0.1201E-15	0.9091E-03	2182.	123.6
25	-0.2414E-15	0.9091E-03	2182.	123.6
26	-0.9835E-16	0.9091E-03	1091.	121.8
27	-0.1638E-15	0.9091E-03	1091.	121.8
28	-0.1922E-15	0.9091E-03	1091.	121.8
29	-0.1297E-15	0.9091E-03	1091.	121.8
30	-0.1946E-15	0.9091E-03	1091.	121.8
31	-0.1003E-15	0.9091E-03	0.0000	120.0
32	0.8982E-18	0.9091E-03	0.0000	120.0
33	-0.4458E-16	0.9091E-03	0.0000	120.0
34	-0.1175E-15	0.9091E-03	0.0000	120.0
35	-0.1801E-15	0.9091E-03	0.0000	120.0

In the second flow problem the two soils stand side-by-side in the permeameter such that the flow through the samples is parallel. The data file flow2.dat is identical to flow1.dat except the for element connectivity portion shown below.

```
KW ELEM
C    4-node connectivity table
C Ele#  Mat #     4 corner nodes           midside nodes
C  left side elements material 1
    1     1     1     2     7     6     0     0     0     0
    2     1     2     3     8     7     0     0     0     0
C  right side elements material 2
    3     2     3     4     9     8     0     0     0     0
    4     2     4     5    10     9     0     0     0     0
    5     1     6     7    12    11     0     0     0     0
    6     1     7     8    13    12     0     0     0     0
    7     2     8     9    14    13     0     0     0     0
    8     2     9    10    15    14     0     0     0     0
    9     1    11    12    17    16     0     0     0     0
   10     1    12    13    18    17     0     0     0     0
   11     2    13    14    19    18     0     0     0     0
   12     2    14    15    20    19     0     0     0     0
   13     1    16    17    22    21     0     0     0     0
   14     1    17    18    23    22     0     0     0     0
   15     2    18    19    24    23     0     0     0     0
   16     2    19    20    25    24     0     0     0     0
   17     1    21    22    27    26     0     0     0     0
   18     1    22    23    28    27     0     0     0     0
   19     2    23    24    29    28     0     0     0     0
   20     2    24    25    30    29     0     0     0     0
   21     1    26    27    32    31     0     0     0     0
```

22	1	27	28	33	32	0	0	0	0
23	2	28	29	34	33	0	0	0	0
24	2	29	30	35	34	0	0	0	0

The flow velocity for the side-by-side soil configuration is the same as for the homogeneous soil case for each soil. That is, the velocity in the fast soil was 10 times the velocity in the slow soil. The velocity calculated for the nodes on the boundary between the soils has a velocity that is the average of the velocities for points within the two soils.

Oscillations in Flow Problems

For certain types of problems, such as those modeling consolidation, oscillations may occur in displacements and pore pressures at the start of the solution, even for steady-state conditions. No simple rule or advice is adequate to avoid these oscillations. The user must be aware that such behavior can take place and that some adjustment of time steps or mesh size may be required to minimize or eliminate oscillations. Typically, shortening the time step duration, increasing the number of substeps, or changing the typical element size may minimize oscillation problems. Trial and correction may be required to produce a stable solution. Initial conditions may influence the potential for oscillation. Methods for establishing satisfactory initial conditions are discussed in the following section on coupled flow.

Coupled Flow-Deformation Problems

STUBBS is capable of solving coupled flow-deformation problems. Loading of soil under drained conditions permits consolidation of the soil, a common coupled flow-deformation analysis. Several example problems in this appendix considered pure deformation of a soil under plain strain and axisymmetric conditions. Three of these problems are presented as coupled flow-deformation examples. The problems are listed in Table C14.

Table C14
Cross Reference of Coupled Flow-Deformation Examples to Pure Deformation Examples

	Pure Deformation	Coupled Flow - Deformation
Plain strain	ate_1.dat	cate_1.dat (top drainage only)
Axisymmetric	ate_ax1.dat	cateax1.dat (top drainage only)
Axisymmetric	ate_ax1.dat	cateaxr.dat (top and radial drainage)

In these problems, the ultimate deformation should be identical to that calculated for the corresponding pure deformation examples. In the pure deformation examples, the deformation is presumed to occur instantaneously, whereas with the coupled flow-deformation, the deformation is constrained to take place as water flows away from the loaded area. In the coupled flow-deformation problems the STUBBS program computes the effective stresses. In the pure deformation mode, the program computes the total stresses because the pore water pressure is set to zero.

The proper establishment of initial conditions is critical to most coupled flow problems. The conditions of concern are the initial pore water pressures and the initial boundary loadings. A-priori determination may not be possible. A pragmatic solution is to apply the proper boundary loading and permit the program to come to equilibrium under these conditions. Then, the loading of interest and the construction sequence can be activated using the construction script. Some programs have a restart feature to permit the initial conditions to be calculated by the program, and then, the program-calculated initial conditions become the starting conditions for the analysis. STUBBS does not have a restart feature. As just mentioned, one possible method for getting the proper initial conditions is to allow the program come to a equilibrium under the known boundary conditions. In the coupled examples, elapsed time up to 1,000 time units is reserved for calculating existing conditions. Then, the elapsed time in excess of 1,000 time units represents "real elapsed time." The deformation at the end of the initiation period must be subtracted from subsequent deformations to obtain the "real" expected values. In the pure deformation BMS example ate_1.dat, the y-displacement of nodes 31 through 35 were calculated. The values of displacements at nodes 31 through 35 are shown in the table for the end of the initial periods (up to Time = 1,000). Table C15 shows the values for the y-displacement after attaining initial equilibrium under the steady state loading, and the values after attaining equilibrium under the applied boundary loading. The net "real" displacement is compared with the pure deformation examples and the hand solution.

Table C15
Comparison of Coupled Flow-Deformation Plain Strain with Hand Solution

Y-displacement for CATE_1.	Node 31	Node 32	Node 33	Node 34	Node 35
End of analysis (T = 6,000)	0.1108	0.1108	0.1109	0.1108	0.1108
End of initiation period (T = 1,000)	0.0042	0.0042	0.0042	0.0042	0.0042
Net "real" displacement	0.1066	0.1066	0.1067	0.1066	0.1066
Hand calculation (also, ATE_1.DAT)	0.1067	0.1067	0.1067	0.1067	0.1067

The **preferred** method is to specify in the proper initial conditions for pore water pressure using the KW IDPP. By using IDPP the displacements that are required to obtain equilibrium under the boundary conditions, body

forces, and boundary loadings can be accounted for in the initial displacement offsets that are calculated within the program. The displacement offsets are stored and are used to establish the origin for displacements. The example files that are provided on disk have the IDPP statements in place. If the IDPP statements are "commented out" or deleted the program will produce the results shown in Tables 15, 16, and 17. If the IDPP lines are used, no initiation period is necessary and the proper answers, the "real displacements" shown in the Tables, will be obtained by time T= 1,000.

Table C16
Comparison of Coupled Flow-Deformation Axisymmetric CATEAX1 with Hand Solution

Y-displacement for CATEAX.	Node 31	Node 32	Node 33	Node 34	Node 35
End of analysis (T = 6,000)	0.1272	0.1272	0.1272	0.1271	0.1271
End of initiation period (T = 1,000)	0.0072	0.0072	0.0072	0.0071	0.0071
Net "real" displacement	0.1200	0.1200	0.1200	0.1200	0.1200
Hand calculation (see also, ATE_AX1.DAT)	0.1200	0.1200	0.1200	0.1200	0.1200

Table C17
Comparison of Coupled Flow-Deformation Axisymmetric CATEAXR with Hand Solution

Y-displacement for CATEAX.	Node 31	Node 32	Node 33	Node 34	Node 35
End of analysis (T = 6,000)	0.1272	0.1272	0.1272	0.1271	0.1271
End of initiation period (T=1,000)	0.0072	0.0072	0.0072	0.0071	0.0071
Net "real" displacement	0.1200	0.1200	0.1200	0.1200	0.1200
Hand calculation (see also, ATE_AX1.DAT)	0.1200	0.1200	0.1200	0.1200	0.1200

As in the case of the flow problems, some oscillations may occur displacements and pore pressures on the way to arriving at steady-state conditions. Again, no simple rule or advice is adequate to avoid these oscillations. The user must be aware that such behavior can take place and that some adjustment of time steps or mesh size may be required to minimize or eliminate oscillations.

Plastic Materials

Plastic material properties can be specified by using the KW ENDO. The parameters entered with ENDO are used in the endochronic model of plasticity developed by Valanis and Peters. A report describing the determination of the endochronic parameters is in preparation. One problem using endochronic parameters is presented to demonstrate the potential of the model to account for plastic deformation. The geometry and loading for the problem is identical to FOURS1.dat except that endochronic soil properties are provided in addition to the linear elastic properties. The data file is named FOURENDO.DAT and is shown below.

```
STUBBs Hand Mesh # 2  4-node elements
C  July 17 ,1995
C  filename = FOURENDO.DAT
C  example using zero unit weight and 10 psi boundary load along top
C  of specimen (axisymmetric, plastic soil)
KW UNIT
C   ATM PRES,  GAMMA H2O,  VALUE OF 1-g  (3) Crit  Npass  Theta
C     psi,        pci,       units of g
      14.7      0.0361         1.0         1E-6      3     0.5
C
KW SPEC
C FLOW, DFORM, DRAIN, CHECK_MESH, CHECK_SCRIPT, AXIS (6)
   F      T      T        F            F           T
C
KW XELT
C  Elements existing at the start of the problem
   1:24
KW SMPR
C  linear elastic material properties
C MAT#,  GAMMAT, RVOID, GS, RSAT (5)
    1     0.00    0.55   2.7   .80
C MODULUS, POISX, POISY, XN, XM, KO (6)
   1000    .3333   .3333   1.0 0.4 0.5
C PERMX, PERMY, B-Value  (3)
  .001    .001    .999
C van Genuchten parameters (2)
   0.006  2.3
KW ENDO    ....Soil properties for plastic response.
C
C  ==================== Foundation Parameters================
C  ================ Clay  1  Su  = 2000 psf  =========
C  M     Fo_H    BETA     Fo_S     A_11
C  Fo_H = stress at reference void ratio
C  BETA = compressibility index
C  Fo_S = cohesive strength
C  A_11 = friction coefficient
C
    1    1.E05    6.3    13.89  0.000001
C
C  Mcv          A_22       Co
C    Mcv = constant volume stress ratio
C   A_22 = Dilatency resistance parameter
C    Co  = Dilatency parameter
   1.1        0.001      0.001
C
C  A1           A2          A3          A4
C  Cyclic response parameters
   0.03       0.740       0.220       0.01
C
C   ALPHA2      ALPHA3        ALPHA4
    3.25E6      1.0E5         100.0
C  End of material  properties
C
KW NODES
    1.0    1.0
    1      0.00          0.00      0      1      0
    2      1.0           0.000     0      1      0
    3      2.0           0.000     1      1      0
    4      3.00          0.000     0      1      0
    5      4.00          0.000     0      1      0
```

```
        6       0.00           2.000     0     0     0
        7       1.000          2.000     0     0     0
        8       2.000          2.000     0     0     0
        9       3.000          2.000     0     0     0
       10       4.000          2.000     0     0     0
       11       0.000          4.000     0     0     0
       12       1.000          4.000     0     0     0
       13       2.000          4.000     0     0     0
       14       3.000          4.000     0     0     0
       15       4.000          4.000     0     0     0
       16       0.000          6.000     0     0     0
       17       1.000          6.000     0     0     0
       18       2.000          6.000     0     0     0
       19       3.000          6.000     0     0     0
       20       4.000          6.000     0     0     0
       21       0.000          8.000     0     0     0
       22       1.000          8.000     0     0     0
       23       2.000          8.000     0     0     0
       24       3.000          8.000     0     0     0
       25       4.000          8.000     0     0     0
       26       0.000         10.000     0     0     0
       27       1.000         10.000     0     0     0
       28       2.000         10.000     0     0     0
       29       3.000         10.000     0     0     0
       30       4.000         10.000     0     0     0
       31       0.000         12.000     0     0     0
       32       1.000         12.000     0     0     0
       33       2.000         12.000     0     0     0
       34       3.000         12.000     0     0     0
       35       4.000         12.000     0     0     0
C
KW ELEM
     1      1      1      2      7      6      0      0      0      0
     2      1      2      3      8      7      0      0      0      0
     3      1      3      4      9      8      0      0      0      0
     4      1      4      5     10      9      0      0      0      0
     5      1      6      7     12     11      0      0      0      0
     6      1      7      8     13     12      0      0      0      0
     7      1      8      9     14     13      0      0      0      0
     8      1      9     10     15     14      0      0      0      0
     9      1     11     12     17     16      0      0      0      0
    10      1     12     13     18     17      0      0      0      0
    11      1     13     14     19     18      0      0      0      0
    12      1     14     15     20     19      0      0      0      0
    13      1     16     17     22     21      0      0      0      0
    14      1     17     18     23     22      0      0      0      0
    15      1     18     19     24     23      0      0      0      0
    16      1     19     20     25     24      0      0      0      0
    17      1     21     22     27     26      0      0      0      0
    18      1     22     23     28     27      0      0      0      0
    19      1     23     24     29     28      0      0      0      0
    20      1     24     25     30     29      0      0      0      0
    21      1     26     27     32     31      0      0      0      0
    22      1     27     28     33     32      0      0      0      0
    23      1     28     29     34     33      0      0      0      0
    24      1     29     30     35     34      0      0      0      0
C
KW LFTC
KW TIME
   10    0.1
KW CSBL
   31  0   32   -10 -10 -10
   32  0   33   -10 -10 -10
   33  0   34   -10 -10 -10
   34  0   35   -10 -10 -10
KW DONE
KW TIME
   10    0.1
KW CSBL
   31  0   32    10   10   10
   32  0   33    10   10   10
   33  0   34    10   10   10
   34  0   35    10   10   10
KW DONE
KW DONE
```

In this problem the column is loaded with a 10-psi column load and then unloaded. If only linear elastic material properties were used the specimen

would return to is original dimensions when the load was removed. With the plastic properties provided by KW ENDO. The specimen has some plastic strain that is observed by the irrecoverable strain (set) present after unloading.

Two versions of FOURENDO.DAT were run for purposes of comparison. One run had KW ENDO and endochronic parameters were supplied. The column of soil was loaded uniaxially with a vertical pressure of 10 psi. In the other run, KW ENDO and the lines containing the endochronic properties were commented out. The results are shown in Table C18.

Table C18
Vertical Displacement and Vertical Stress

Data file	Loading	Node 31	Node 35	Sigma Y
with ENDO	10 psi	0.1201	0.1201	10.0
without ENDO	10 psi	0.1200	0.1200	10.0
with ENDO	unloaded (0 psi)	0.0027	0.0027	<E-5
without ENDO	unloaded (0 psi)	<E-15	<E-15	<E-14

A small amount of set is calculated when ENDO is used. A very small amount of residual stress remains in the specimen when ENDO is used.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*

2. REPORT DATE
March 1997

3. REPORT TYPE AND DATES COVERED
Final report

4. TITLE AND SUBTITLE
User's Manual for STUBBS: A Finite Element Program for Geotechnical Analysis

5. FUNDING NUMBERS

6. AUTHOR(S)
John F. Peters, Ronald E. Wahl, Ronald B. Meade

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
U.S. Army Engineer Waterways Experiment Station
3909 Halls Ferry Road
Vicksburg, MS 39180-6199

8. PERFORMING ORGANIZATION REPORT NUMBER
Instruction Report GL-97-1

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
U.S. Army Corps of Engineers
Washington, DC 20314-1000

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES
Available from National Technical Information Service, 5285 Port Royal Road, Springfield, VA 22161.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution is unlimited.

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

STUBBS is two-dimensional finite element code developed for the purpose of solving problems in geotechnical engineering. The code is capable of performing a fully coupled Biot analysis (Biot 1941) in which the partial differential equations for static equilibrium and flow continuity are solved simultaneously. The code can also handle either pure deformation or pure flow problems. The code accounts for nonlinear stress-strain behavior of soils and groundwater flow through both saturated and partially saturated soils. The program is designed to account for soil-structure interaction effects using a library of specialized elements. Thus, STUBBS has the potential to deal with soil-structure interaction, consolidation, and transient seepage problems. The program can perform analysis of problems in either plane strain or axisymmetric geometries. The computer program was written in FORTRAN77 and runs successfully on several platforms that include personal computers, engineering work stations, and mainframe computers.

A particular strength of the program is the ability to simulate construction processes. As such, the program can predict movement,s displacements, and porewater pressures that might exist at different times during construction. Activities that can be simulated by STUBBS include the excavation and placement of fill, the addition/removal of structural elements such as walls, anchors, or bracing, and the placement and removal of distributed and concentrated loads. Analysis of the the

(Continued)

14. SUBJECT TERMS

Computer program	Finite elements
Consolidation	Seepage
Construction modeling	Soil-structure interaction

15. NUMBER OF PAGES
113

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	UNCLASSIFIED		

13. (Concluded).

construction process enables the design engineer to evaluate the effects of construction rates and the sequence of the construction activities. Typical applications of this feature include the construction of embankment dams, lock walls, reinforced earth projects, and sheet-pile walls. This simulation of construction activities is controlled by the use of a construction script.